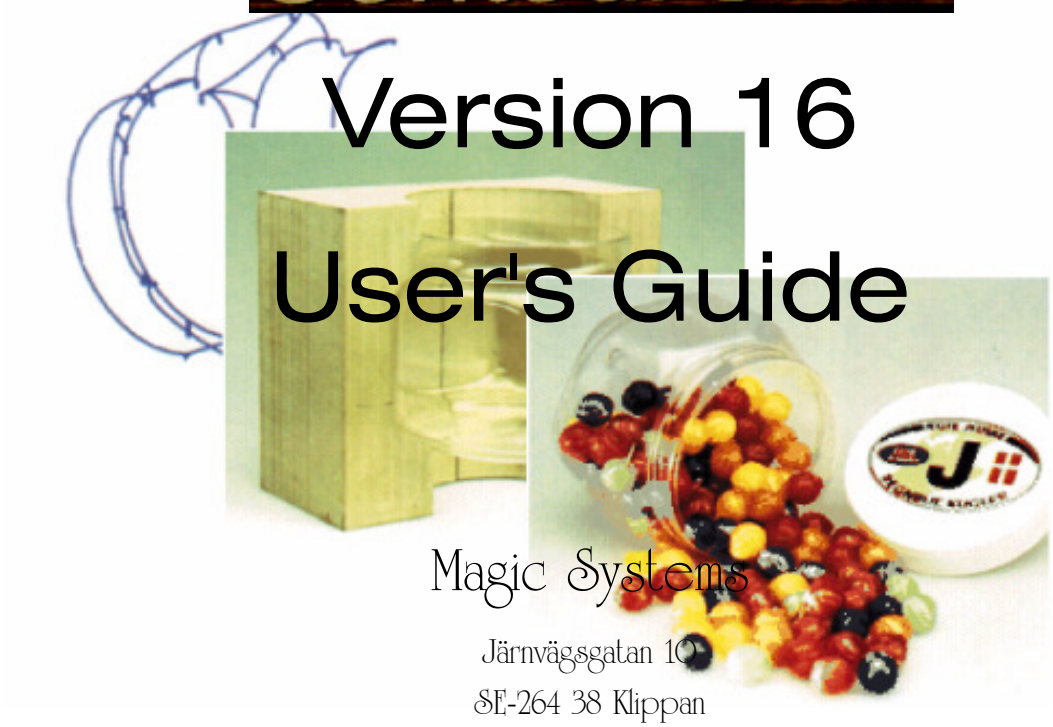
ContourCAM

Version 16

User's Guide



Magic Systems

Järnvägsgatan 10

SE-264 38 Klippan

Tel: +46 (0)435 44 81 81

www.magicsystems.com

www.contourcam.com

The material presented in this publication is copyright-protected © 1997 - 2005 by Magic Systems and may not be reproduced in any form, by any method for any purpose without the prior written consent of Magic Systems. Information in this document is subject to change without notice. It is against the law to copy the software, except in accordance with the terms of the licensing agreement.

PROGRAM LICENSE AGREEMENT

IMPORTANT NOTICE: Please read the terms of the following License Agreement carefully. Your use of the product signifies your acceptance of the terms of the Agreement. If you do not agree with the terms of this Agreement, you should promptly return the package; your money will be refunded. Retain this License Agreement for future reference.

Article 1: License Grant

The software is the intellectual property of Magic Systems and its licensors, and is protected by law, including Swedish copyright laws and international treaties.

Magic Systems grants to you a license:

1. To use the program on a single machine.

2. To make a single archival back-up copy of the program in support of your use of the single program on a single machine.

3. To modify the program and/or merge it into another program for use on a single machine.

4. To transfer the program to another party if that party agrees to accept the terms and conditions of this Agreement, and you do not retain any copies of the program, whether in printed, machine-readable, modified, or merged form. Except as expressly provided for in this license, you may not copy, modify, or transfer this program.

5. If the software in this package is licensed as a Site License, it is licensed for use on several machines. Additional master copies of the software may be made by the licensee, equal to the number of licenses purchased.

No reverse engineering. Except as stated above, YOU MAY NOT MODIFY, TRANSLATE, DISASSEMBLE, OR DECOMPILE THE SOFTWARE, OR ANY COPY, IN WHOLE OR PART.

Article 2: Term

The license is effective until terminated. You may terminate the license at any time by destroying the program together with all copies, modifications, and merged portions in any form. The license will also terminate upon conditions set forth elsewhere in this Agreement, and Magic Systems may terminate your license if you fail to comply with this Agreement. You agree, upon such termination for any reason, to destroy the program together with all copies, modifications, and merged portions in any form.

Article 3: Disclaimer of Warranties and Limitations of Remedies

1. Magic Systems software is licensed to you As Is. You, the consumer, bear the entire risk relating to the quality and performance of the software. In no event will Magic Systems be liable for direct, indirect, incidental, or consequential damages resulting from any defect in the software, even if Magic Systems had reason to know of the possibility of such damage. If the software proves to have defects, you, and not Magic Systems, assume the cost of any necessary service or repair. In the event any liability is imposed on Magic Systems, Magic Systems' liability to you or any third party shall not exceed the purchase price paid for this product.

2. Thirty-day limited warranty on disks. Magic Systems warrants the enclosed disks to be free of defects in material and workmanship under normal use for 30 days after purchase. During the 30-day period, you may return a defective disk to

Magic Systems with proof of purchase, and it will be replaced without charge, unless the disk is damaged by accident or misuse.

Replacement of a disk is your sole remedy in the event of a defect. This warranty gives you specific legal rights, and you may also have other rights which vary from state to state.

3. THE LIMITED WARRANTY STATED ABOVE IS THE ONLY WARRANTY OR REPRESENTATION OF ANY KIND WITH RESPECT TO THE SOFTWARE MADE BY Magic Systems OR ANY THIRD PARTY INVOLVED IN THE CREATION, PRODUCTION, DELIVERY, OR LICENSING OF THE SOFTWARE. Magic Systems AND ANY SUCH THIRD PARTY DISCLAIM ALL WARRANTIES, EXPRESS OR IMPLIED, WITH RESPECT TO THE SOFTWARE, ITS QUALITY, RELIABILITY OR PERFORMANCE; OR ITS MERCHANTABILITY, NON-INFRINGEMENT OF THIRD PARTY RIGHTS, OR FITNESS FOR A PARTICULAR PURPOSE.

4. Some states do not allow the exclusion or limitation of implied warranties of liability for incidental or consequential damages, so the above limitations or exclusions may not apply to you.

Article 4. Product Serial Number and Required Activation:

Many of the software programs Magic Systems produces contain a serial number and activation system designed to prevent unlicensed or illegal use of the software. If this software contains such a system you agree that Magic Systems may use these measures to protect Magic Systems against software piracy. Software using this system may contain technology that limits the ability to install and uninstall the software on a computer to not more than a finite number of times for a finite number of computers. This License and the software containing this technology require activation as further set forth in the Installation program of the software. The software using this system will only operate for a finite period of time prior to software activation by you. During activation, you will be asked to provide your unique product serial number provided with the software over the Internet to verify the authenticity of the software. If you do not complete the activation within the finite period of time as prompted by the software, the software will cease to function until activation is complete, which will restore software functionality. In the event that you are not able to activate the software over the Internet, you may contact Magic Systems Customer Service

via the telephone to complete the activation process, using the information provided by Magic Systems during activation, or as may be set forth in the documentation.

Article 5: General

1. You may not sub-license, assign, or transfer the license or the program except as expressly provided in this Agreement.

Any attempt otherwise to sub-license, assign, or transfer any of the rights, duties, or obligations hereunder is void.

2. This Agreement will be governed by the laws of the Sweden, and you agree that any claims regarding the software shall be brought in Sweden, and waive any objections to jurisdiction in the Swedish Courts.

3. You acknowledge that Swedish laws prohibit the export/re-export of technical data of Swedish origin, including software, and agree that you will not export or re-export the software without the appropriate Swedish and foreign government license.

COPYRIGHT

ContourCAM is copyright 1997 - 2005 Magic Systems

DesignCAD 3D MAX, DesignCAD Express and the IMSI logo are registered trademarks of IMSI.

Adobe, Acrobat, and the Acrobat logo are trademarks of Adobe Systems Incorporated, which may be registered in certain jurisdictions.

Flash is a registered trademark of Macromedia.

Introduction

Getting Started

Introduction

ContourCAM is a powerful CAM addition to DesignCAD 3D MAX. DesignCAD is developed purely as a drawing system and has been given several awards.

Because of the incredible capacity, and because it is so user friendly, we chose to base ContourCAM on DesignCAD 3D MAX to achieve our objectives: A CAD/CAM system, which lacks nothing in capacity, which is easy to learn and use, and extremely quick to convert a drawing into NC code.

We believe that we have achieved our objectives, and we are confident that, when you have worked with the system, you will agree.

System Requirements

To use ContourCAM, you must have, at a minimum, the following hardware and software installed in your computer:

- Approximately 70 megabytes of hard drive space for a complete installation.
- DesignCAD 3D MAX ver. 16
- Microsoft Windows 2000, or Windows XP.

Although not required to run ContourCAM, the following hardware is recommended for better performance:

- Pentium processor.
- at least 256 megabytes of RAM.
- SuperVGA graphics card capable of 256 or more colors.
- SuperVGA monitor capable of at least 800x600 resolution.

Installing ContourCAM

For your convenience, the installation of ContourCAM is automated. You simply insert the CD-rom into your CD-rom drive and the installation process will start automatically.

If not, please follow the instructions below.

Manually Installing ContourCAM

1. Select Start in Windows.
2. Choose the Run command.
3. Type `D:\AUTORUN.EXE` and either press ENTER or click OK in the Command Line field.
4. Follow the instructions on the screen.

If you use a CD-ROM drive other than D:, replace that drive letter in the command line instructions above.

ContourCAM 2D and 3D

ContourCAM consists of an editor, a two dimensional and a three dimensional system.

Usually it's both quicker and easier to create drawings in two dimensions rather than in three, and this is fully exploited in ContourCAM, since by using simple directions you can prepare three dimensional objects from two dimensional drawings.

You can of course create 3D objects by constructing them in 3D as well.

ContourCAM Setup

Once you have installed ContourCAM on top of DesignCAD 3D MAX, you will probably be raring to start testing it out, but before you can actually start turning out products or prototypes on your CNC device, you will have to configure ContourCAM thoroughly. Not only will you need to closely consult the next chapter, ContourCAM Configuration, but you will need the manual for your machinery as well.

Sources of Information About ContourCAM

You can learn about ContourCAM in various ways. The supplementary learning tools are: the tutorial in Chapters 7 and 8 of this manual, the online Help system, and the sample drawings. We recommend that you read the manual thoroughly, so you don't miss any of the details that make ContourCAM exceptional.

The User's Guide

The User's Guide consists of 6 chapters describing the functionallity of ContourCAM, giving examples of the various possibilities. Some of the examples you will find as samples in the \ContourCAM Samples Folder. We also recommend that you go through chapters 7 and 8, which will guide you through some examples of how ContourCAM works. Chapter 7 concerns 2D CAM plus a number of excellent drawing and editing additions and chapter 8 is about 3D CAM.

The Online Help System

ContourCAM offers a complete, online Help system. You can access it from the Help command on the Main menu. In addition, context sensitive help lets you press F1 anywhere within the program for help on that area.

The Sample Drawings

ContourCAM comes with a number of sample drawings located in the \ContourCAM Samples subdirectory. These drawings include examples of the various facilities that ContourCAM offers as well as files for the lessons of the tutorials Chapters 7 and 8.

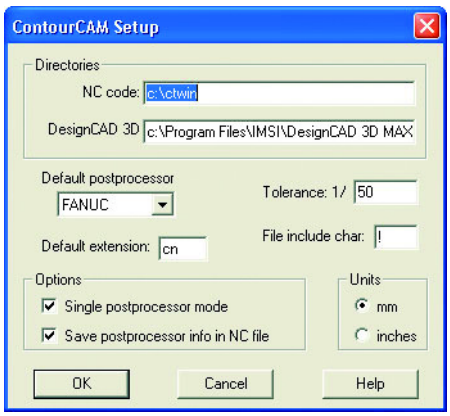
We hope you will enjoy your new software.

1 ContourCAM Configuration

In this chapter the program element **ContourCAM Configuration** is described. Here the general setup of ContourCAM takes place. Also here postprocessors are created and edited. Finally, the machine cycle library will be updated from this program.

1.1 General settings

By activating **General** in the menu **Setup** the dialog below appears.

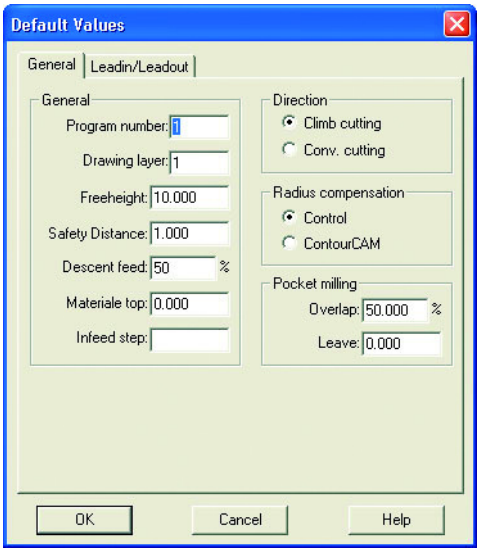


- **NC code**: Here you enter in which directory NC programs should be placed by default. This can be overwritten in the postprocessors.
- **DesignCAD 3D**: Here you enter, where DesignCAD 3D MAX is installed.
- **Default postprocessor**: Here you enter which postprocessor will be preselected in commands that require the choice of a postprocessor.
- **Default extension**: Here you can enter an extension that automatically is added to file names of NC programs for which there is no extension specified.
- **Tolerance**: As the NC code is generated a path through a given geometry is found. If the geometry has small gabs between some entities the tolerance can be set to for instance 1/200 with the result that the geometry will be regarded as unbroken provided that the gab between two entities is 0.005 mm or less. (1/200=0.005).
- **File include character**: When the NC program is transmitted from the editor to an external unit (e.g. a CNC unit) the editor will read files to be included in the NC program. In this field you enter which sign in the NC code indicates that the following text is a file name which is to be included when transmitting. Normally '!' is used, but some times it can be necessary to choose another. Notice that files which are to be included must be entered on a line of their own.

- **Single postprocessor mode**: If this option is checked ContourCAM will not ask you to select a postprocessor when working with files without an attached postprocessor but the default postprocessor will always be assumed.
- **Save postprocessor info in NC file**: If this option is checked ContourCAM will not store information of attached postprocessor in the NC files. This means that when you load file again it will only have a postprocessor if the **Single postprocessor mode** is checked or the NC file is placed in a directory with a default postprocessor, see Section 1.6.
- **Units**: You can choose between millimeters and inches. This only affects how output is presented.

1.2 Default values

By activating **Defaults** in the **Setup** menu the dialog below appears. In this dialog you can alter the values used to initialize the dialogs for defining NC processes.



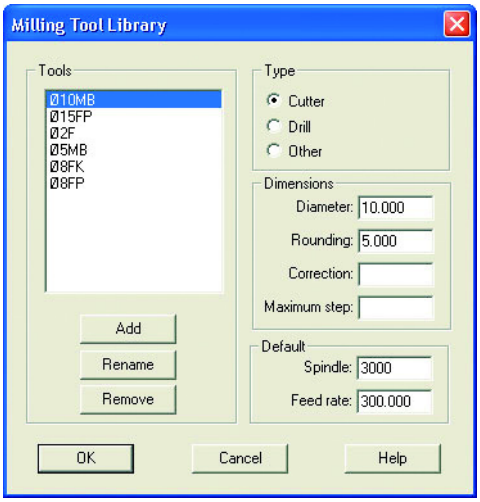
- **Descent feed**: When the default option is checked in tool selections the descent feed will be this percentage of the general feed rate.
- **Material**: Default value for top of material.
- **Infeed step**: Default value for infeed step.
- **Pocketing**: Default values only related to pocket milling.
- **Lead in**: The default settings for lead ins.
- **Direction**: Choose between climb and conventional milling.
- **Radius compensation**: The default radius compensation.
- **Lead out**: The default settings for lead outs.

1.3 Tool Library

ContourCAM has two tool library; one for milling machines and one for lathes.

1.3.1 Tools for milling machines

When you activate the milling tool library the dialog below appears:

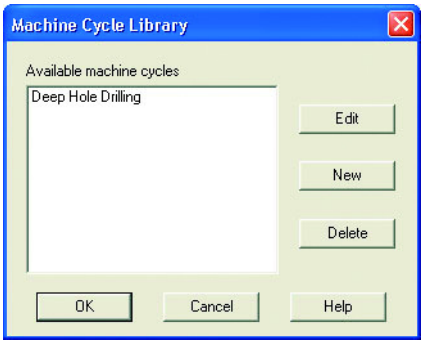


- **Tools**: Here you have the list of all the tool currently in the library. The tool that is highlighted is the one currently being edited.
- **Add**: Pressing this button lets you add a tool to the library. Enter the name of the tool you wish to add (use a name which makes the tool easy to recognize)
- **Type**: What type of tool.
- **Dimensions**: The contents of this group depends on the type of the tool.
- **Diameter**: The diameter of the current tool, i.e. twice the radius.
- **Rounding**: Enter a value between 0 (if the tool is flat) and the tool's radius (if the tool is ball faced).
- **Displacement**: Here you can enter the value of length compensation for this tool. Often this value will be stored in the control so you may leave this field empty.
- **Maximum step**: If you enter a value here, ContourCAM will prevent you from using a deeper infeed step with this tool.
- **Spindle**: Enter here the default spindle speed for this tool.
- **Feed rate**: Enter here the default feed rate for this tool.

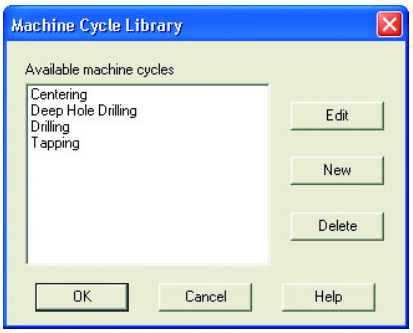
1.4 Machine cycle library

The machine cycle library is activated from the **postprocessor** menu. By this you get an alphabetical list of the defined machine cycles. When a machine cycle is highlighted it can be edited or deleted. Furthermore, new cycles can be defined.

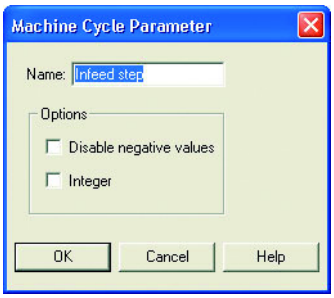
For editing and setting up you will have the same dialog which will be described in the following.



1.4.1 Editing machine cycles



In the machine cycle library you only define the general interface for the machine cycles, i.e. the name and parameters. What NC code is generated for each cycle is defined is the various postprocessors, thus you can use machine cycles without considering which CNC unit you are going to use. Please note that the machine cycles are not referred to internally by their names hence changing names will not have undesired side-effects.



It is possible to define up to 10 parameters for each machine cycle. When a cycle is employed, the user will be asked to enter values for each parameter.

When a parameter is defined or edited the dialog in the figure appears. Also the name of the parameters can be changed without influencing the current values.

You can avoid negative values to be entered by activating **No negative values**. If you activate **Integer** values in the NC code will be written without decimals.

1.5 Postprocessors

Postprocessors are very important in ContourCAM since they describe to ContourCAM how a specific CNC machine will be controlled.

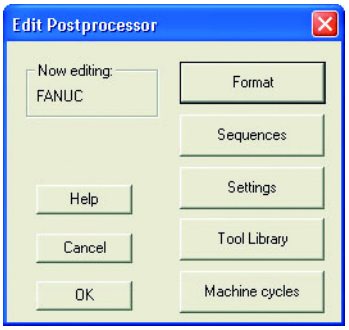
In this section we will describe the dialogs, where you configure your postprocessors. While reading this try to consider how the postprocessors for your CNC machines should be configured.

With ContourCAM there are several examples of postprocessors for more common controls - as an inspiration for your own postprocessors - you can try and see, how they are configured.

When you set up or edit a postprocessor a dialog box like this one will appear. Each of the five big buttons activates a new dialog box where the configuration of the postprocessor takes place.

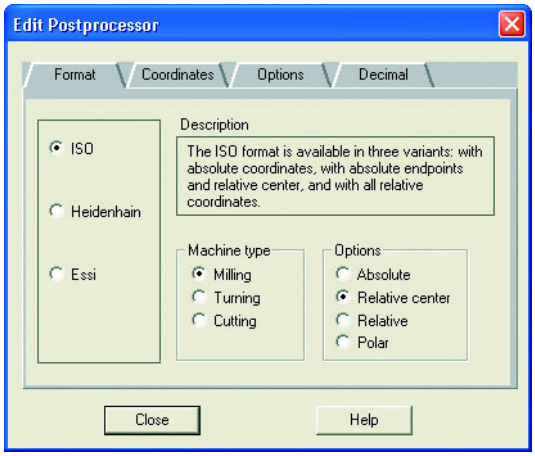
Some of the boxes contain tabs with several pages.

In the following the dialog boxes for configuring postprocessors will be described.



1.5.1 Postprocessor format

When the format of the postprocessor is to be stated a dialog box containing 4 tabs appears. The following sections will describe each tab separately.



1.5.1.1 Format-tab

Under this tab the general format of the NC code is selected.

•**ISO**: Sets the general NC format to be ISO, which means G codes.

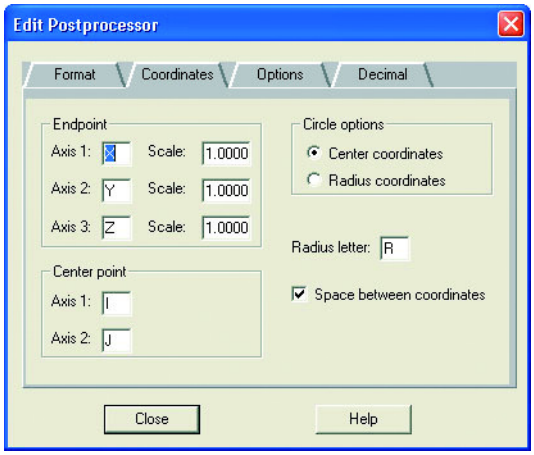
•**Heidenhain**: Sets the general NC format for Heidenhain controls.

•**Essi**: Adjustment to ESSI-controls.

•**Milling machine**: Indicates that the CNC machine is a milling machine. Therefore up and down movements between different milling tasks will be generated automatically.

- **Turning machine**: Indicates that the machine is a Turning machine.
- **Cutting machine**: Indicates that it is a flame/laser or plasma cutting machine.
- **Absolute**: All coordinates are specified from the origin.
- **Relative center**: Centers of circles are specified relative to the current position endpoints from the origin. Only available with ISO.
- **Relative**: All coordinates are specified relative to the current position of the tool.
- **Polar**: All coordinates are stated as polar with radius in the first axis and the angle in the second axis. Only available with ISO.

1.5.1.2 Coordinate-tab

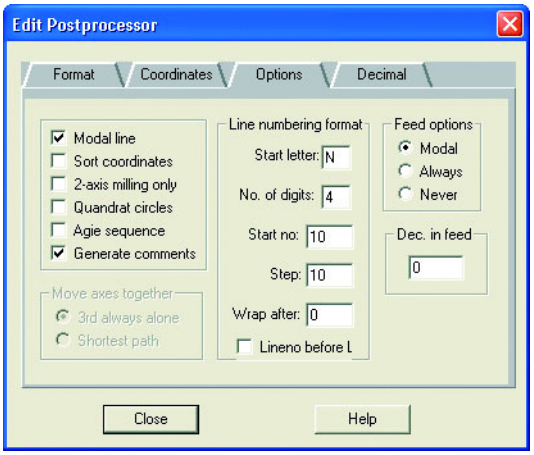


In this tab adjustments of coordinates are placed.

- **Endpoint|Axis 1**: Here you enter the character for the 1. axis normally X.
- **Endpoint|Axis 2**: Here you enter the character for the 2. axis normally Y.
- **Endpoint|Axis 3**: Here you enter the character for the 3. axis normally Z.
- **Endpoint|Scale**: For each coordinate you enter a factor which the coordinates in your drawing will be multiplied by before written in the NC code. Normally, the factor will be 1.
- **Center point|Axis 1**: Here you enter the character for the 1. axis of the center coordinates normally I.
- **Center point|Axis 2**: Here you enter the character for the 2. axis of the center coordinates normally J.
- **Center coordinates**: If center coordinates is selected arcs will always be written with a specified center. This will not influence reading of NC code. Only ISO format.
- **Radius coordinates**: If radius coordinates is selected arcs not representing a full circle will be written using a radius statement.
- **Radius letter**: Character for radius statements.
- **Space between coordinates**: States if spaces between coordinates and other code should be added.

For Turning machines it is often necessary to change the end points and center point fields from their default values. A Turning machine working with diameters the first axis must be named Z with a scaling of 1 and the second axis must be named X with a scaling of 2. Study the construction of the postprocessor for OKUMA.

1.5.1.3 Options-tab



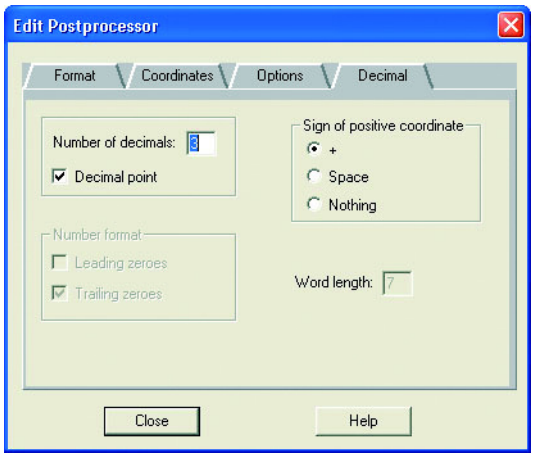
In the tab **Options** you find fields for adjustment of different elements common for most postprocessors.

- **Modal line**: Selecting Modal line implies that the coordinates will be generated modally, i.e. if a coordinate does not change during the movement, then this coordinate will not be written in the NC code. Also the G code for linear movement is only written, if the previous was not a line. The use of modal lines can make the NC code shorter, especially in connection with 3D programming
- **Sort coordinates**: Selecting Sort coordinates implies that the coordinates in the NC code will be sorted alphabetically (Endpoints and center coordinates separately). Otherwise they will be written sorted according to their axis number. Normally, coordinates are sorted by the axis, but lathes for instance may require that the coordinates be sorted alphabetically.
- **2-Axis milling only**: 2-Axis milling only is used when a milling machine can only control two axes simultaneously, ContourCAM will then generate all movements in all 3 axes as 2 movements. If 2-axis milling only has been chosen then the Move axes together field is enabled.
- **Quadrant circles**: Quadrant circles are used, if a CNC machine only accepts arcs within a quadrant, ContourCAM will then divide arcs passing through more than one quadrant into a number of arcs.
- **Agie sequence**: States that the NC code is meant for an AGIE erosion machine. The reason, why an AGIE machine must be treated in a special way, is that the NC code does not come in the same order as the preparation path. If you have an AGIE erosion machine you ought to be aware that the simulation of AGIE programs reflects the exchange of lines in the NC code. The example of a postprocessor for AGIE shows the configuration of an AGIE.
- **Move axes together**: This field is used to describe how simultaneously movements with three axes are to be generated for CNC machines only able to move two axes at the same time. This field is only selectable if **2-axis milling** is active. **3. always alone** tells that the third axis must always be moved alone; meaning, if the movement of the third axis is negative, then axis 1 and 2 are moved first followed by the third and the opposite if the third axis is moved positive. By **Shortest path** the combination of axes leading to the shortest movement is chosen. This possibility is used if the machine is able to work with two arbitrary axes.
- **Start letter**: States the letter preceding the line number, which normally

is the letter N. If it is a Heidenhain control, there will be no start character.

- **Number of digits**: No of digits states how many digits the line number should contain. If No of digits for instance is 4 then the line number 10 will be written as N0010, while 0 as No of digits states that only the necessary number of digits should be written.
- **Start number**: Start number states which line number to start with.
- **Step**: Step states the step between succeeding line numbers.
- **Wrap after**: If the control has a maximum allowed line number it can be stated here. When generating NC code the line number count will be reset when the indicated number is exceeded.
- **Lineno before L**: If this option is not checked N numbers are not added to lines starting with L. This only applies to the ISO format.
- **Feed options**: This field states how ContourCAM should specify feed rates in the NC code. **Modal** states that the feed rate should be written only when it changes. **Always** states that feed rate must be written with every movement in the NC code. This is specially used by older Heidenhain controls. **Never** states that feed rates will never be included in the NC code.
- **Dec. in feed**: Here you specify with how many decimals the feed rate should be written in the NC code. This only influences the output format (NC code and simulator).

1.5.1.4 Decimal-tab



In the decimal tab you state how decimal-numbers are written in the NC code.

- **Number of decimals**: In this field you enter the number of digits to put after the decimal point. Newer controls normally use 3 decimals while older controls often have a precision of 2 decimals.
- **Decimal point**: Indicates whether numbers are written using a decimal point - this is normal for newer controllers - or a fixed format. If Decimal point is not selected the **Number format** field is enabled.
- **Sign of positive coordinate**: In this field you configure how positive coordinates are written. Choosing + implies that positive coordinates start with a +, e.g. X+100. Choosing Space implies that positive coordinates start with a space, e.g. X 100. While Nothing implies that no sign precedes positive coordinates, e.g. X100
- **Number format**: This field is only enabled if Decimal point not has been selected. In this field you configure how decimal numbers without a

decimal point are represented. Selecting **Leading zeroes** implies that decimal numbers are written with zeroes in front of them, the **Word length** field will become selectable. Selecting **Trailing zeroes** implies that decimal numbers are written with pending zeroes.

- **Word length**: This field is only enabled, if **Leading zeroes** is checked in the **Number format** field. In this field you enter the word length of decimal numbers. The word length is necessary to be able to write decimal numbers with leading zeroes. Using 2 decimals and a word length of 6 the coordinate X100. is written as X01 if you use leading zeroes only, if you use both leading and trailing zero X100. is written as X010000 and as X10000 using trailing zeroes only.

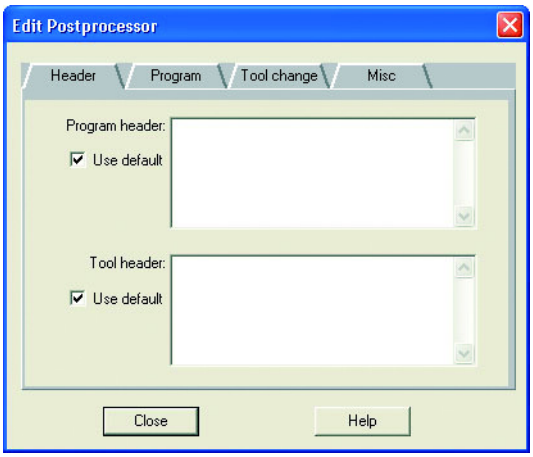
1.5.2 Sequences in the postprocessor

To set standardized parts of NC programs code sequences are used. You state a fixed pattern of NC code containingvariables. ContourCAM places a number of system variables at your disposal. When generating NC code the system variables are replaced by their values. The system variables is written in braces, {}, case is ignored. The following system variables are supported:

- {PNO} Program number.
- {X} 1. coordinate.
- {Y} 2. coordinate.
- {Z} 3. coordinate.
- {CX} 1. center coordinate.
- {CY} 2. center coordinate.
- {FED} Feed rate.
- {SPN} Spindle feed.
- {TNO} Tool number.
- {TIR} Index for radius compensation.
- {TIL} Index for length compensation.
- {NTL} Next tool.
- {TNM} Tool name.
- {TRD} Tool radius.
- {TLN} Tool length.
- {FRH} Free height.
- {TX} Tool-X-measure (only Turning machines)
- {TZ} Tool-Z-measure (only Turning machines)
- {DATE} Current date.
- {1},{2},... Parameters for machine cycles.

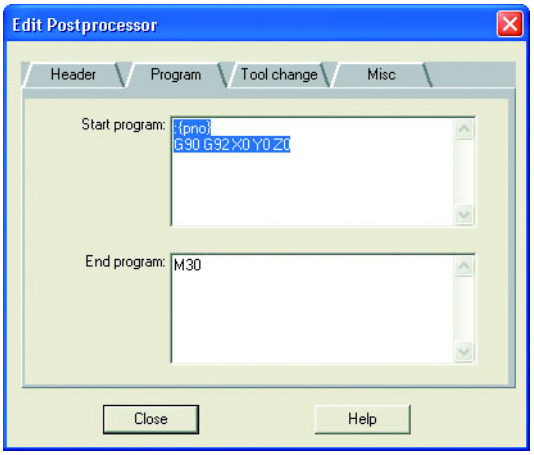
Like the format dialog the sequence dialog contains a number of tabs which will be described separately in the following.

1.5.2.1 Header-tab



- **Header**: Here you can enter a text that is inserted in the top of the NC program before the actual NC code but after the default header. The default header is included only if the field **Use default** is checked. An example of a default head is shown below.
- **Tool header**: Here you can enter a sequence that is inserted for each tool used in the NC program. These sequences will be included directly after the program start and are typically for Heidenhain to state TOOL DEF.

```
' This NC program was generated by ContourCAM
'
' Version: 16.0
' Program: Toolpath
' Date:  July 23, 2005
' Drawing: C:\Program Files\IMSI\DesignCAD 3D MAX 16\5PLATE.DCD
' Postpro: FANUC
'
```

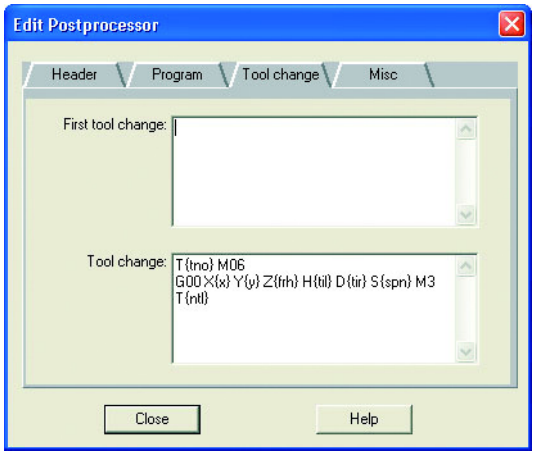


1.5.2.2 Program-tab

In this tab you enter the sequences for the beginning and ending of NC programs respectively.

•**Start program**: This sequence typically contains the program number and various setup codes.

•**End program**: Here you state codes telling the control that the program ends.



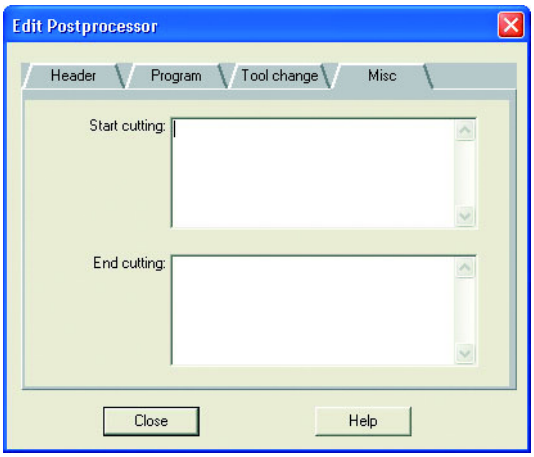
1.5.2.3 Tool change

Here you state the codes that are used for tool change. Of course the tool number will be included but also other variables like spindle, indices to the compensation table, positioning, next tool etc.

It is possible to specify a special sequence for first tool change in **First tool change**. If this field is empty the sequence entered in the field **Tool change** is used for all tool changes.

1.5.2.4 Misc.-tab

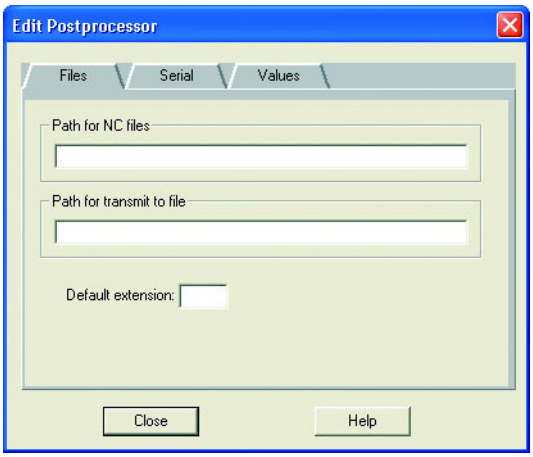
In the tab **Misc.** sequences that are included immediately before and after the toolpaths are entered. Often for milling they will be empty but if using for instance a plasma cutter it is used for turning on and off the torch.



1.5.3 Settings

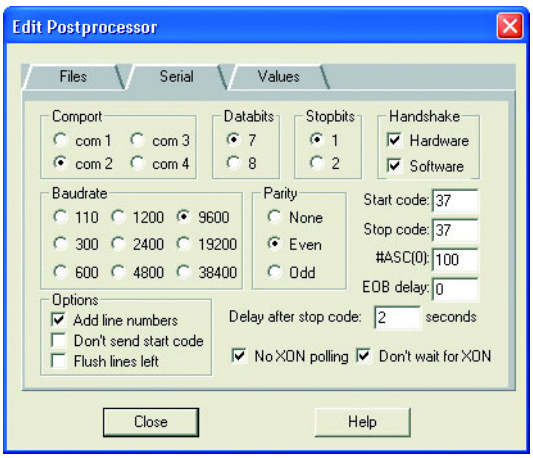
The dialog that appears when clicking settings has three tabs.

1.5.3.1 Files-tab



Here you can state directories and file extensions to NC filesspecific to this postprocessor. If nothing is stated the values entered in the general setup of ContourCAM are used.

- **Path for NC files**: If you want the NC programs for this postprocessor placed in another directory than the one stated under ContourCAM configuration, you can indicate it here.
- **Path for 'Transmit to file'**: Here you state where the result of 'Transmit to file' is to be saved. If you state nothing it is saved in the default directory for NC programs.
- **Default extension**: If you want to have a special extension added to the NC files for this postprocessor by default, you can specify it here.



1.5.3.2 Serial-tab

Under the **Serial** tab the parameters concerning communication with the CNC unit via RS232 are adjusted. The fields in this dialog must be adjusted so they are consistent with the similar parameters on the control.

•**Comport**: In this field you choose which serial port on the computer to use for the transmission to the controller. When using COM3 and COM4, you may run into problems. The normal settings are: for COM3 base address 3E8h, interrupt 0Ch, interrupt level 4 is normally used and for COM4 base address 2E8h, interrupt 0Bh, interrupt level 3. The problem with these settings is that COM3 and COM4 will share interrupt and interrupt level with COM1 and COM2 respectively. The result is that if a mouse is connected to COM1 then after using COM3 the mouse will no longer work. If you want to use another setting you will have to change the hardware (normally some jumpers must be adjusted) and in the Windows setup.

- **Databits**: In this field you choose the number of data bit used. 7 data bit is the normal when communicating with CNC machines while

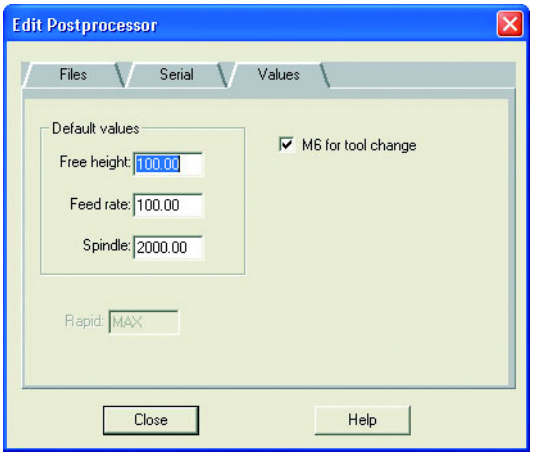
communication between two computers normally uses 8 data bit.

- **Stopbits**: In this field you choose the number of stop bits. 1 stop bit is the usual but with communication speeds lower than 600 baud 2 stop bits are sometimes used.
- **Handshake**: In this field you choose which type of handshake you want. If you choose not to use handshake ContourCAM will start transmission without waiting for the machine to become ready. Similarly, when receiving ContourCAM will not try to stop the CNC machine, if data are send too fast. Both cases can result in loss of data. If you want to use DNC transmission, some sort of handshake must be chosen. Hardware handshake is also called RTS/CTS handshake. With hardware handshake the computer and CNC machine need two extra connections in addition to the ones need to the communication. Using these two connection both the computer and CNC machine can stop the other if the transmission goes too fast. The other possibility is Software handshake, this is also called XON/XOFF or DC codes. Using software handshake the communication wires are used by the receiver to start and stop the transmitter. To decide which sort of handshake and how to connect it refer to the manual for the CNC machine.
- **Baud rate**: In this field you choose the communication speed. If you do not use handshake then you should not choose a speed higher than 2400 baud. If you use handshake speeds up to 38400 baud are possible.
- **Parity**: In this field you choose which parity to use. Even parity is the normal when using 7 data bit and None when using 8 data bit. Odd parity is rarely used.
- **Start code and Stop code**: In the Start and Stop code fields you can enter special ASCII codes, which are transmitted before and after the transmission of the NC code. These codes are required by some controls to indicate the start and stop of the actual NC code. For instance most Heidenhain controllers uses stop code 3. If you enter 0 as start or stop code no code will be transmitted. When receiving it means that possible noise before and after the transmission will not result in strange characters in the NC program.
- **#ASC(0)**: In this field you enter how many ASCII code 0 ContourCAM shall transmit after the NC code. This imitates empty tape on a tape reader and is demanded by some older controllers.
- **EOB delay**: Amount of nulls to send after end-of-block, i.e. after each line. Some older controls requires time to process each line.
- **Add line numbers**: Check if you want automatic line numbering during transmission to the CNC unit.
- **Don't send start code**: Check this if the start code should not be send to the CNC unit. The start code will then only be used to determine program start on reception.
- **Delay after stop code**: When ContourCAM receives a stop code from the controller it is interpreted as end of transmission. However some controllers raise the alarm if the connection is closed immediately. Therefore it is possible to state a delay of a number of seconds so the controller has time to finish.
- **No XON polling**: If you use only software handshake, when it is ready

to receive, ContourCAM will continuously send an XON signal to the CNC unit every 2 seconds until it starts receiving data. Check this option to prevent ContourCAM from sending XON's before data is received.

- **Don't wait for XON**: If you use software handshake ContourCAM will not begin transmitting before it receives an XON signal. Check this option to start the transmission immediately.

1.5.3.3 Value-tap



Under the tab **Values** standard values that are used when generating NC code are stated.

•**Free height**: This field contains the default safety height of the machine. When milling this height is used for positioning by default.

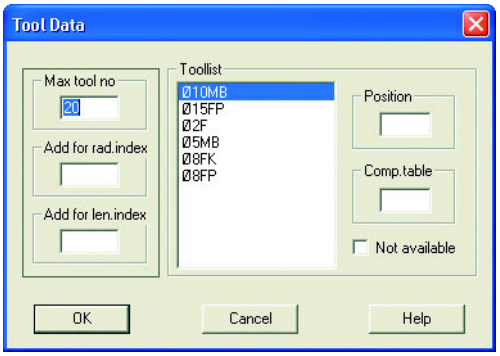
•**Feed rate**: This field contains the default feed rate. This is used for both NC generation and simulation.

•**Spindle**: This field contains the default spindle value.

- **Rapid traverse**: In this field you enter which text specifying the rapid traverse movements on a Heidenhain (normally MAX or 9999). This field is enabled only if Heidenhain is selected in the Format-tab.

1.5.4 Tool library

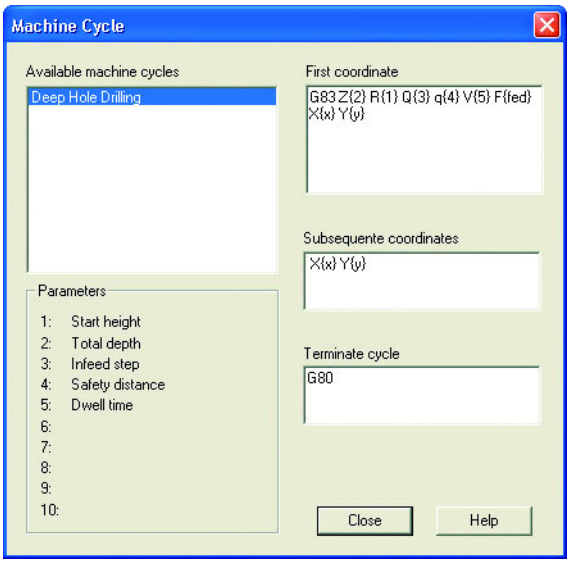
When activating the tool library the dialog below appears.



- **Max tool number**: Here you state the highest tool number allowed. ContourCAM will not assign tool numbers higher than this number.
- **Add rad. index**: On ISO controls values for radius and length compensation are typically stored in tables. Often you choose the tool number as index to the table, but if it is not possible to save radius and length values in the same index, you can choose to add a fixed offset to the tool number for one of the values. If you want to place the values of the radius compensation with an offset you can enter the offset here.

- **Add length index**: Here you can enter an offset for the length compensation index.
- **Tool list**: This list shows all the tools you have defined for the current type of postprocessor.
- **Position**: Here you can enter the position for the current tool. If you enter a number other than zero this will be the tool number used for this tool. If you leave this field empty, ContourCAM automatically assigns a position/number, when the tool is used.
- **Not available**: Checking this option will prevent this tool from being used with this postprocessor.

1.5.5 Machine cycles



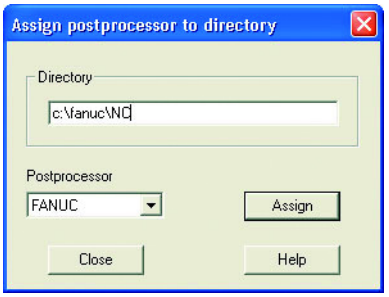
Under machine cycles you define, how the current post-processor generates code for the machine cycles defined in the machine cycle library see section 1.4.

- **Machine cycles available**: Here you choose the machine cycle to be changed. Notice that the contents of the remaining fields are changed, when a new machine cycle is selected, but the changes for all machine cycles are stored.
- **Parameter**: Here parameters defined for the current machine cycle are shown.
- **First coordinate**: Here you enter the NC code for initiating the machine cycle including the first coordinate. You can refer to the values of the parameters using the system variables {1}, {2} etc. To get the coordinates {X} and {Y}are used, in addition all other system variables can be used, see section 0.
- **Subsequent coordinates**: Here you enter the code sequence for the remaining coordinates.
- **Terminate cycle**: Here possible NC codes that end the use of the machine cycle can be entered.

1.6 Assign default postprocessor

In ContourCAM you can assign a postprocessor to be default in a directory. This means that all files in that directory by default will have this postprocessor attached. You can assign a postprocessor to any number of directories, but there can be only one postprocessor assigned to each directory. If you assign two postprocessors to the same directory only the last one will count.

You activate this command from the post-processor menu. A dialog as shown appears.

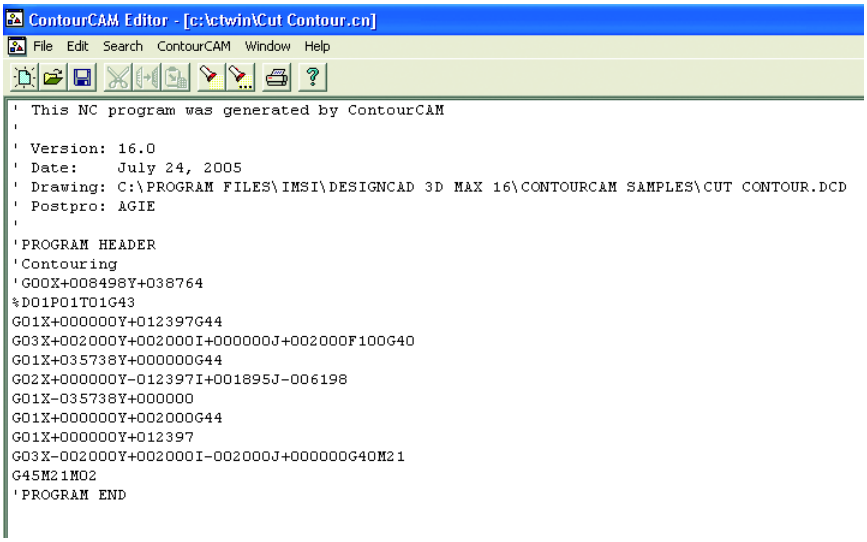


- **Directory**: The directory to be assigned the postprocessor.
- **Postprocessor**: The postprocessor you wish to assign to the given directory.
- **Assign**: When you press this button a file indicating the default postprocessor for the entered directory.

ContourCAM Editor

2

The ContourCAM editor forms the connection between ContourCAM and your CNC machines. It is in the editor that you can make changes in the NC code programs, simulate NC code on-screen and communicate with the CNC machines.



The main part of the editor is, as the name implies, a text editor. It works as most other text editors. The size of the files you can edit is limited only by the amount of memory Windows offers. And since Windows uses the hard drive as an expansion of the memory, you can work with file sizes of several mega-bytes of memory.

As long you have enough memory there is no limit to the number of files that can be read into the editor. The first nine files will be numbered from one to nine, and you can quickly switch between them using the menu **Window**.

2.1 Editing files

When starting the editor there are no files in the editor. You can either read a file using Files|Open (see section 2.4) or begin a new file using **Files|New**.

2.1.1 Short-cuts while editing

The following is a list of short-cuts and their function in the ContourCAM editor. A sequence of keys with a plus (+) in between means all the keys should be typed simultaneously. For example <Shift>+<Ctrl>+R <Shift> means you should hold down both <Shift> and <Ctrl> while pressing R. If a sequence of keystrokes are separated by spaces it means you press them down after each other.

For example <Ctrl>+K R means you should press both <Ctrl> and K, release them, and then press R. Finally a sequence of keystrokes separated by a comma means they have the same function.

Key	**Means**
<↑>, <↓>, <→>, <←>	Move cursor
<Ctrl> + <→>	Move cursor one word right
<Ctrl> + <←>	Move cursor one word left
<Page uo>	Move one page up
<Page down>	Move one page down
<End>	Move cursor to the end of the line
<Home>	Move cursor to the beginning of the line
<Ctrl> + <Home>	Move cursor to the top of the file
<Ctrl> + >End>	Move cursor to the end of the file
>Ctrl> + G	Go to line number
<Insert>	Toggle between Insert/Override
<Delete>	Delete the character underneath the cursor
<Backspace>	Delete the character to the left of the cursor
<Ctrl> + Y	Delete line
<Ctrl> + Q Y	Delete till the end of the line
<Shift> + cursor move-ment	Mark text
<Ctrl> + K B	Begin marking (block)
<Ctrl> + K Y	Delete marked text (block)
<Ctrl> + X	Cut marked text (block) to clipboard

Key	Means
<Ctrl> + C	Copy marked text (block) to clipboard
<Ctrl> + V	Insert from clipboard
<Ctrl> + S	Search
<Ctrl> + R	Search and replace
<Ctrl> + E	Search and exchange
<F3>	Repeat the last search, exchange or replace

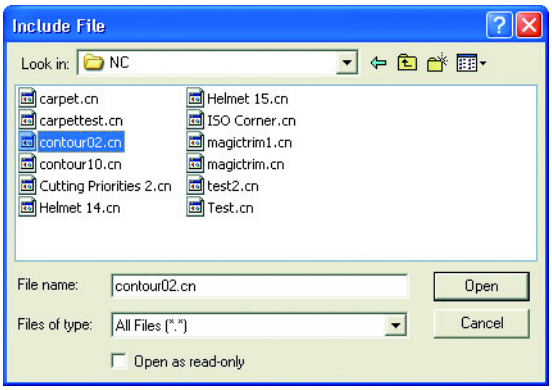
2.1.2 Hot-keys in ContourCAM Editor

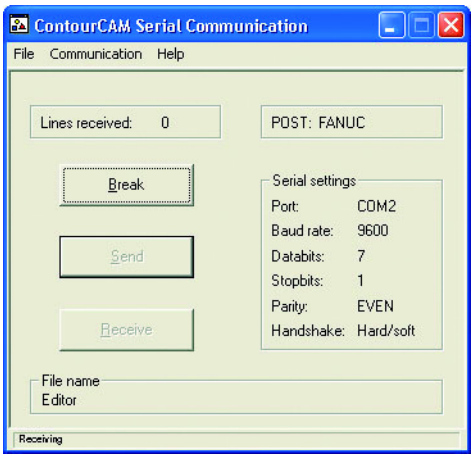
In addition to the short-cuts while editing there are several hot-keys to quickly activating commands in the menus.

Key	Meaning
<F1>	Help
<Ctrl> <F12>	Read program from disk
<F12>	Save program to disk
<Shift> + <Ctrl> + R	Insert file at cursor position
<Shift> + <Ctrl> + W	Save marked text (block) to disk
<Alt> + <F4>	Exit editor
<F5>	Receive program
<F6>	Transmit program
<Ctrl> + P	Print program
<F7>	Change to visual simulation of NC code

2.1.3 Include file

The command **Include file** can be found in the **Edit** menu. If you have a file on your hard drive that you would like to include in your editor, you select include file in the menu. Select the file in the directory and the file will be inserted at the cursors position.





2.1.4 Include from Serial

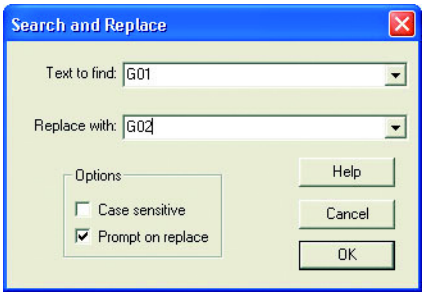
The command **Include from Serial** is located in the **Edit** menu. This is one of two ways to receive via the serial port. In this way the received data will be inserted at the cursors position in the actual window. The editor will be locked until the serial transmission has ended, either by itself or by the user. For more detail see section 2.4.1.

2.2 Search functions

As in any other editor you can also search and replace texts in the ContourCAM editor. You also have the option to swap two different texts. However, it is not possible to search for texts that exceeds one line.

The dialogs for the three search functions search, replace and swap, are made on the same principle so only replace will be discussed.

This illustration shows the dialog that pops up when the replace function is activated.



- **Find**: The text to search for is entered here.
- **Replace**: The text to replaced with is entered here.
- **Case sensitive**: Check if you want case sensitive search.
- **Prompt on replace**: If this field is checked you will be asked to confirm each replace.

Please note that if you have selected a block when activating the replace function, it will be performed on the selected block only. Otherwise it will be performed from the cursor position to the end of the text

Besides searching for texts you can also search for patterns. A pattern consist of normal text and the following special characters:

* : indicates whole words until a space or a tabulator.

\# : numbers, including sign (+,- and space)

_ : white space, space or tabulator.

\ : used when searching for special characters. For example * \# _ \\

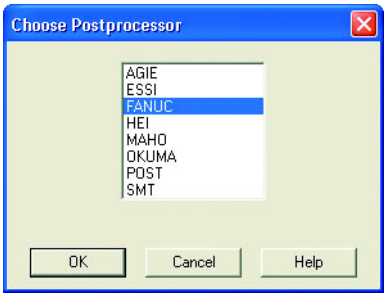
For example X# will match X10, X-20.7, X.56 etc. The pattern T* will match all words starting with T.

2.3 CAM-functions

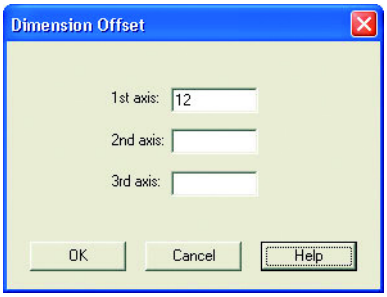
Especially the commands in the **CAM** menu makes the ContourCAM editor special. Most of these commands demand that there is a file with an associated postprocessor in the active text window.

2.3.1 Choose postprocessor

If there is no postprocessor associated to the file or you would like to associate another postprocessor to the file, you can select this command. When the command is activated, a list containing all the post-processors in the system will be displayed. You can select the new postprocessor you would like to associate to your file.



2.3.2 Dimension offset



This command makes it possible to offset the whole or parts of the program along one or more axes. You will be asked to enter the displacement for all three dimensions. If the program was written in ISO-code the lines after a G91 until the next G90 will not be offset.

2.3.3 Remove Line Numbers

This command removes all line numbers. Normally, you work without line numbers in ContourCAM. The line numbers will not be appended, until the transmission to a CNC unit starts.

2.3.4 Add Line Numbers

If you are using other software than ContourCAM for the transmission and you do not wish to use 'transmit to file', this function can be use to add line numbers to the NC program.

2.3.5 Insert start-/stop sequences

Normally the start- and stop sequences indicated in the postprocessor will be inserted, when the NC code is generated. However, it is possible to insert start- and stop sequences in the editor.

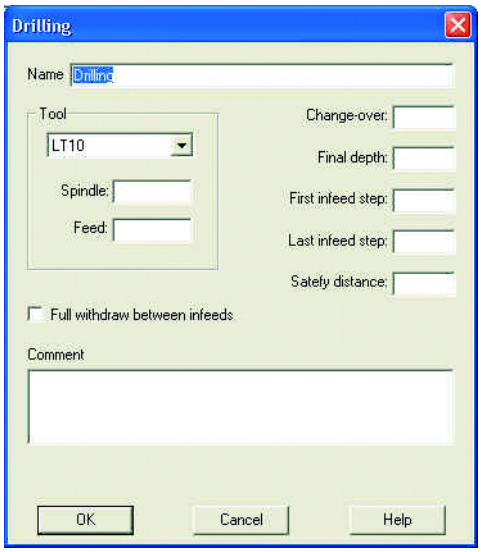
2.3.6 Drilling on lathes

Most controllers on new turning machines contain machine cycles to for deep hole boring. However, there are still many older controls without this facility.

In ContourCAM there is a command to generate NC code for deep hole boring on a lathe.

Before activating the command a window in the Editor must be open and also a postprocessor must be selected. When the command is activated a dialog as shown in the figure appears.

- **Change-over point**: Here you enter the point where the feed rate should start.
- **Final depth**: The final depth of the hole.
- **First infeed**: The initial infeed depth. The sign is ignored. **Last infeed:** The final infeed depth. If a value is entered it must be smaller (numerically) than the initial infeed depth. ContourCAM will calculate the intervening infeed depths so they gradually decrease and the final will be approximately as entered. If nothing is entered the same infeed depth will be used during the whole process.



- **Safety distance**: Here you enter the distance from the material where the program switches from rapid traverse to feed rate.
- **Full withdraw between infeeds**: If you check this option the drill will be moved to the change-over point between each infeed. Otherwise the drill will only be lifted the safety distance between each infeed.

When you entered the values the NC program generated will be placed on the position of the cursor.

2.3.7 Mirror in X

This command is used to mirror an NC-program round the X-axis. This means that negative x-values becomes positive and positive x values become negative. The command has no dialog since it is very simple. If you want to mirror a program round the x-axis, where for instance x=10, this is done by the following sequence:

- Use the command Mirror X.
- Use the command Offset and offset the program with the value 10 along the x-axis.

The program has now been mirrored round the x-axis with x=10.

2.3.8 Mirror Y

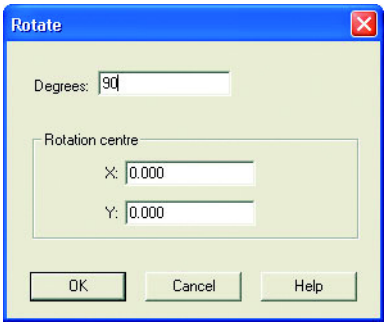
This command is used to mirror an NC-program round the y-axis. This means that negative y-values becomes positive and positive y-values becomes negative. The command has no dialog since the function is so simple.

If you want to mirror a program round the y-axis with y=20, this is done by the following sequence:

- Use the command Mirror Y.
- Use the command Offset and offset the program 20 along the y-axis.

The program has now been mirrored round the y-axis with y=20.

2.3.9 Rotate



This command is used to rotate an NC-program a number of degrees round a stated point. This command has a dialog looks like this one:

•**Degrees**: Enter how many degrees the program must be rotated. A positive number for clockwise rotation and a negative number for counter clockwise rotation.

•**X**: Enter the x-coordinate for the center of the rotation. Standard value = 0.

•**Y**: Enter the y-coordinate for the center of the rotation. Standard value = 0.

Please note that a rotation round 0,0 of 180 degrees is not the same as mirror in Y.

2.4 Calling external programs

Serial communication and simulation of NC programs take place in two external programs. To make it easier for the user two short-cuts have been made to these programs. These shortcuts can be found in the **CAM** menu.

2.4.1 Receive program

Activating this command enables you to receive a program from the control. Serial Communication is activated and it then waits for data to receive.

If you switch back to the editor (or another program) Serial Communication continue in the background. When Serial Communication has received a whole program a new window containing the received program will be opened in the editor. For more details see chapter 6.

2.4.2 Send program

This command activates Serial Communication which will immediately start sending the contents of the active window in the editor. Transmission will also run in the background. For more details see chapter 3.

2.4.3 Send to file

This command's functionality is the same as **Send program** except is saves the program in a file instead of sending it via the serial port.

2.4.4 Simulate NC program

This command start the simulator and let it work on a copy of the program in the active window. For a complete description of NC simulating see chapter 4.

2.5 Loop programming

Solving several simple problematic with a certain similarity can be very trivial. By using loop programming you can repeat a simple procedure several times, without having to write all the coordinates yourself.

Using loop programming, a part of NC code can be repeated a number of times with a linear displacement. Loop programming is indicated using an asterisk (*) followed by the number of repetitions and the coordinate displacement after each repetition.

Hereafter the program follows, terminated by another asterisk (*). It is possible to create nested loops. This could be useful to make an array of holes.

An example of loop programming. To the left the loop code and to the right the result.

When the loop program is written select **Expand loop** in the **CAM** menu. A new window containing the result will be opened. The original program will still exist in the editor. If the result is not, what you expected, you can just close the window and the window with the original loop program will automatically appear.

If ContourCAM locates an error in the loop program, an error message will be displayed on the screen and the cursor will be placed at the error.

It is possible to do displacement along all three axis.

***Note**: There should be a postprocessor assigned to the loop program. If not, use **Choose postprocessor** in the **CAM** menu.*

By nesting loops it is possible to make a grate. The following program makes a grate with 4*6 holes in it:

```
*4 Y10
*6 X10
X0 Y0
 *
 *
```

```
X+0. Y+0.
X+10. Y+0.
X+20. Y+0.
X+30. Y+0.
X+40. Y+0.
X+50. Y+0.
X+0. Y+10.
X+10. Y+10.
X+20. Y+10.
X+30. Y+10.
X+40. Y+10.
X+50. Y+10.
X+0. Y+20.
X+10. Y+20.
X+20. Y+20.
X+30. Y+20.
X+40. Y+20.
X+50. Y+20.
X+0. Y+30.
X+10. Y+30.
X+20. Y+30.
X+30. Y+30.
X+40. Y+30.
X+50. Y+30.
```

Since loops are unfolded from the inside this results in first a row with holes in Y=0 and X in intervals of 10, starting in 0 and ending in 50. In this example each row starts in the left side. To make each second row start in the right side the program can be changed to the following:

```
*2 Y20
*6 X10
X0 Y0
 *
 *6 X-10
 X50 Y10
 *
 *
*6 P60
X100 Y0
*
X100. Y0.
X50. Y86.603
X-50. Y86.603
X-100. Y0.
X-50. Y-86.603
X50. Y-86.603
```

Example of loop programming with rotation. The point (100,0) is being rotated 360 degrees in intervals of 60 degrees.

In addition to displacement you can also do rotations around origin (0,0). This is done by writing the angle (noted by a P) instead of the displacement.

During each iteration of the loop the NC program will be rotated the specified number of degrees.

Rotation is counter clockwise. To achieve clockwise rotation specify a negative angle. Rotation works on both absolute and relative coordinates.

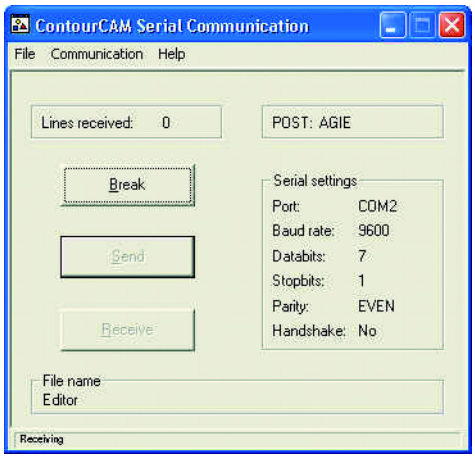
Note: When doing loop programming with rotation it is important to specify both X and Y coordinates for each movement since only the specified coordinates are updated. A modal program will therefore not be rotated correctly.

---

3 Serial Communication

When NC programs are to be transmitted or received from a CNC Unit the program Serial Communication is used. It can operate as an independent program or controlled by the editor. In this chapter the first mentioned method is primary described; for serial communication via the editor see section 2.4

.



3.1 Reception from serial port

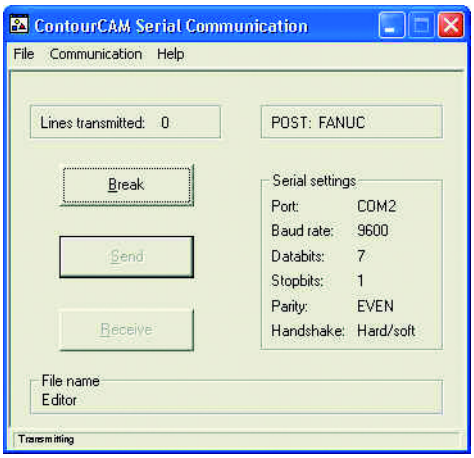
When the program is started the standard postprocessor will automatically be loaded. If you want to receive using another postprocessor you must choose it from the **Communication** menu.

After this receiving is activated either by pressing the receive button or by choosing Receive in the menu

Communication. Before the reception can start you must enter a filename and where you want the received to be saved.

If a start-code is entered in the postprocessor, all data up to and including the start-code will not be saved. Also if a stop code is stated all data received after the stop-code will not be saved (the stop-code itself will not be saved either)

.



3.2 Transmission via serial port

Transmission to an external unit is activated either by pressing the transmit button or by choosing **Transmit** from the **Communication** menu. If a file is no loaded (using **Files|Open**) you will be asked to enter the filename of what is to be transmitted. If software-handshake is used data will not be transmitted, until the ready signal (XON) is received from the external unit. However transmission can be activated by pressing send again.

3.3 Transmission to a file

If you choose Transmit to file from the menu Communication the result will be saved in a file instead of being transmitted to the serial port. This can be useful, if ContourCAM is not used for serial communication as comments are removed and line numbers can be added. Start and stop codes and also ending ASCII 0 signs will not be saved in the file. In the postprocessor you can specify a folder, where the files are to be placed.

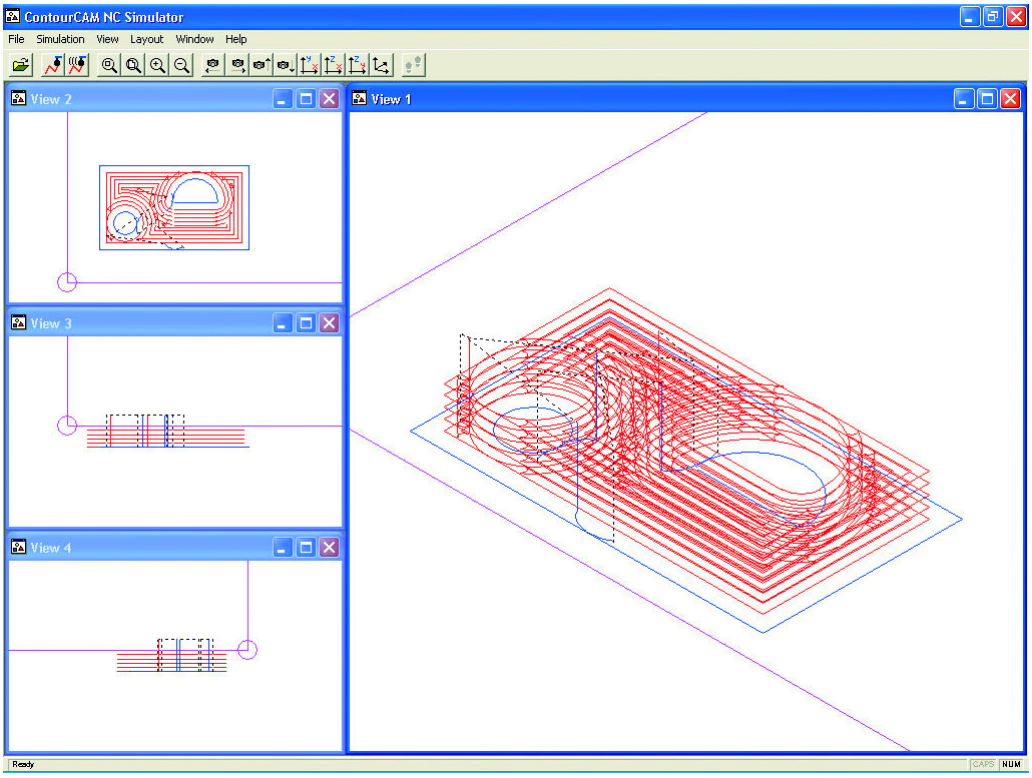
NC Simulator

4

To verify NC programs ContourCAM contains an NC simulator. The simulator displays a graphical image of the NC program. Depending on the NC program the simulator behaves in different ways.

The NC simulator enables you to simulate toolpaths on the screen before sending the NC program to your CNC unit.

If you open it from within the editor, you will get a simulation of the current NC program, but you can even open an NC program from the file menu and have it simulated.



Menus: The NC menu of the simulator includes the following:

File=>Open: Let's you open the file that you want simulated. This file must be a NC program.

File=>Exit: Terminates the program.

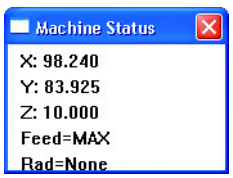
Simulation=>Trace: Starts simulation. If for any reason the simulation has been stopped, this starts the simulation again.

Simulation=>Single step: Starts simulation in single step mode. If, for any reason, the simulation has been stopped, this starts the simulation again. When stepping you can follow each NC-code closely. This is very useful when you want to follow the program, and spot where the presumed error occurs.

Simulation=>Restart: This restarts the current NC program. If for example you have been simulating a program and want to start all over again, this can be achieved in one of two ways: Open the same program again or select this entry in the menu.

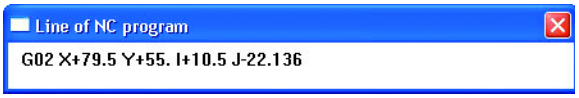
Simulation=>Clear: This clears the simulation windows.

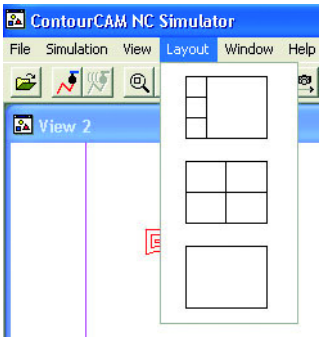
View=>Machine status: This shows the machine status at the current point of the simulation.



View=>Info: This shows length cut, rapid movement, and work time at the current point in the simulation.

View=>NC line: This shows the NC line at the current point of the simulation.





Layout: You can choose between these layouts:

ContourCAM 2D 5

In this chapter we will describe 2D CAM programming using ContourCAM. This consists of a number of commands added to DesignCAD. These extensions make it possible to generate and read NC code and expands DesignCAD's drawing facilities.

Be aware that layers 50 and 254 shouldn't be used since some of ContourCAM's commands internally uses those layers.

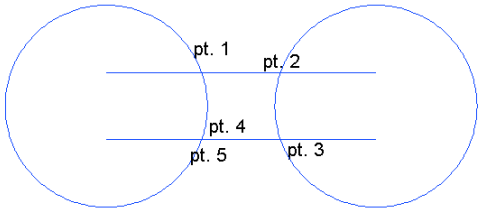
All ContourCAM's CAM commands can be executed using the ContourCAM menu, which the installation will add to your DesignCAD menus. You will also find some extra programs and macros that we have placed in their appropriate DesignCAD menus. In the following we shall indicate, where you will find the program or macro at the beginning of each section.

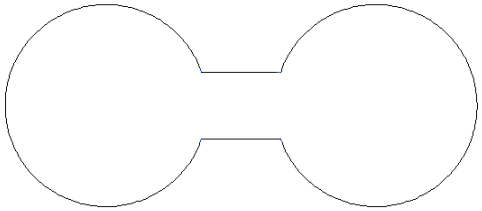
All the special commands Magic Systems has developed as a supplement to DesignCAD's commands will be discussed in the following. In the beginning of the tutorials (chapters 7 and 8) of this manual you will also find examples for most of the commands.

5.1 Editing commands

Editing commands are used for editing already existing entities or create new entities based on already existing entities. To make it easier using DesignCAD for editing NC programs we have added several new commands that will be discussed in this section. Under each command the menu in which it can be found is listed.

5.1.1 MagicTrim: Trimming more elements





Menu: Edit=>Trim/fillet

When using this command you should point out a path through a number of elements. When you have pointed out the path and press <Enter> MagicTrim will automatically trim all the elements so all the elements that are not part of the path are deleted.

When two subsequent entities have two intersections you should always set the point closest to the desired intersection, when pointing out the first entity of the two. A rule of thumb is to always set the points as close as possible to the intersection with the following entity. If you are marking a closed path you should finish by setting a point on the first entity.

Note that you can include several parts of an entity in your path. See section 7.1.5 for an example of MagicTrim.

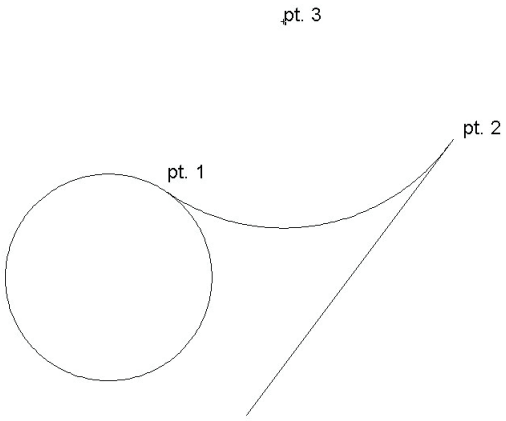
MagicTrim has a lack at present though - it cannot automatically trim polylines. If you select a polyline, when using MagicTrim the command will only be executed until the polyline, after which it stops!

5.1.2 Fillet-2: Draw an arc tangent to line/circle

Menu: Edit=>Trim/fillet

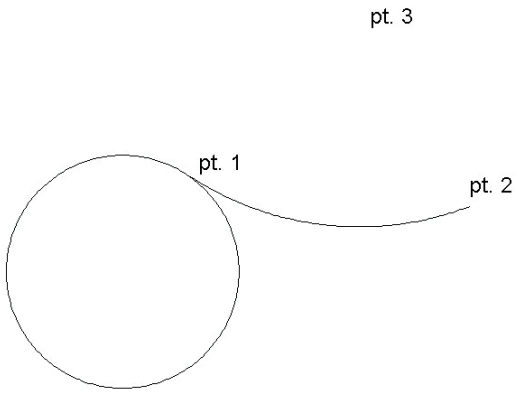
This command makes fillets between circles, arcs and lines. When you activate the command you must set two points close to the entities between which you want the fillet. If the fillet is between circles/arcs it is important to set the points on the correct side of the circles/arcs.

When you have set the two tangent points you must set a point in the general direction where the fillet should be. Finally, the radius of the fillet is entered.



An arc will now be drawn, counter-clockwise, from the first tangent to the other.

5.1.3 Fillet to point: Draws an arc through a point tangent to line/circle



Menu: Edit=>Trim/fillet

This command draws an arc from a point to a line/circle. The arc drawn will be tangent to the line/circle.

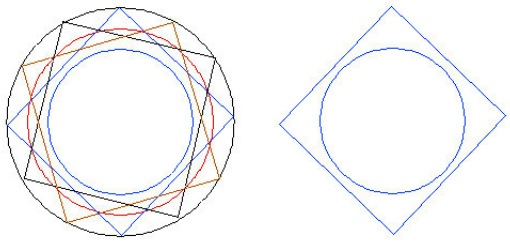
When activating this command you should set a point close to the line/circle that the arc should be tangent to. Next set a point, indicating where the arc should start, and a point in the direction of the center of the arc. Finally, enter the radius of the arc.

Fillet to point will now draw the short arc being tangent to the line/circle you have specified and with center close the point you specified.

5.1.4 Delete all but selected color

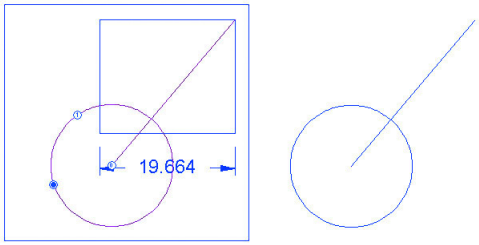
Menu: Edit=>Magic Deletes

This command is very useful if you receive a drawing from a customer or the drawing office stored on a disk. Usually colors are used to separate elements on a drawing. For example one color for text and another for geometries. Typically only one color is used for the toolpaths.



To delete everything on the drawing except elements with a specific color, simply set a point close to a element having the color you want to keep in the drawing.

5.1.5 Delete but selected: Delete all except what is selected



Menu: Edit=>Magic Deletes

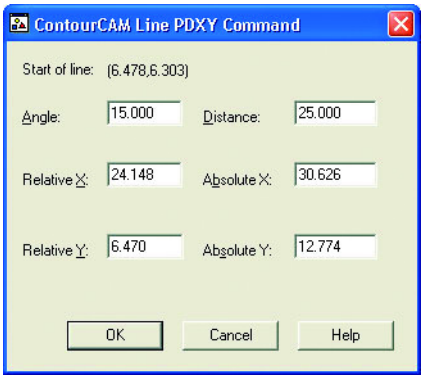
When activating this command you are requested to select the elements you want to keep. When pressing enter all other elements are being deleted.

5.2 Drawing commands

Drawing commands are used when creating new entities on a drawing. Most of our additions supplement and extend existing DesignCAD commands.

5.2.1 LinePDXY: Relative line

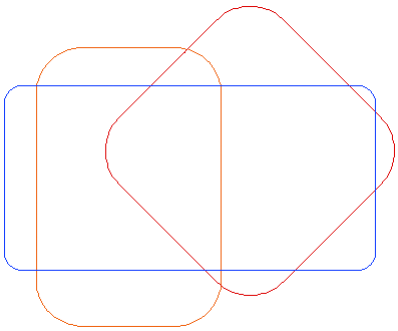
Menu: Draw=>Lines



This command performs a triangle calculation based on two of the following values.

1. P(polar angle)
2. D(polar distance)
3. DX(relative distance along the X-axis)
4. X(absolute X-coordinate)
5. DY(relative distance along the Y-axis)
6. Y(absolute Y-coordinate)

When D (polar distance) and DX(relative distance along the X-axis) or D and DY(relative distance along the y-axis) are used, there are always two solutions. If D is negative the second solution is chosen.



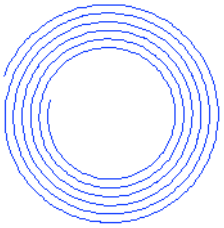
5.2.2 Box2: Draw squares

Menu: Draw=>Planes

This command is designed for drawing one or more squares with rounded corners. The command lets you specify the values of the square. You must enter width, height, the rounding on the corners (0 if they should be sharp) and finally the angle of the square. Next you must point out the center of the square.

If you set points before activating the command these will be use when placing the square.

5.2.3 Spiral: Draw a spiral



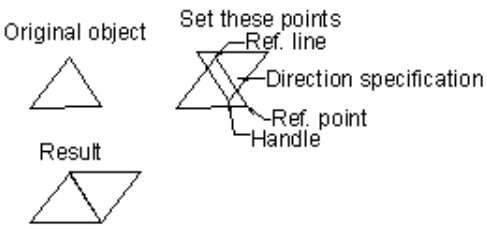
Menu: Draw=>Circles

This command draws a spiral. You must set three points: center, start and stop. When the points are set you are asked how many extra turns the spiral should have. Press <Enter> if none.

If the start point is closer to the center than the stop point the spiral will be counter-clockwise. If the stop point is closer to the center than the start point the spiral will be clockwise.

5.3 Nest commands

These commands are used for repeating a geometry on a drawing. The commands are useful when preparing a drawing for cutting. All the commands described in this section can be found in the **ContourCAM** menu.



5.3.1 Nest-1: Nest one geometry to itself

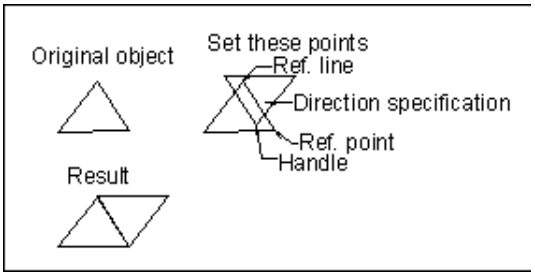
This command is used to nest an object with itself.

First you must set two points used to enclose the object. Then you set a point on the copy for handles, then another point on the original as a reference point for the handle. Next

you must set a point on the reference line and yet another as direction specification for offset (press <Enter> if you do not want a offset). Finally the offset distance is entered.

The copy of the object will be placed with the specified offset parallel to the reference line.

5.3.2 Nest-2: Nest two geometries to each other



This command is used when nesting two different objects against each other.

First select the entity/entities and press <Enter>. Then set a point as reference on the object you have selected. Then another point (reference) on the line you wish to use as reference line. Now set a point on the “target” which refers to the point on the selected object, then another point on the line that should refer to the line you selected on the subject. Set a point for direction of the offset and finally enter the distance for the offset.

The selected object will be straightened to its reference parallel to the reference line on the “target” with the two reference points straightened to each other.

5.4 2D NC programming

In this section we describe ContourCAM’s facilities for supporting CAM programming in DesignCAD. The philosophy behind ContourCAM is to make the generated NC code, based on a drawing, as complete as possible.

In a CAD program a drawing consist of some drawing elements (lines, circles, text etc.) called entities. When generating NC code based on a drawing one has to arrange the entities in a structured way, so that the generated code is constructed in a reasonable way. Typically a drawing can be read in more than one way. The programmer can, based upon his knowledge concerning the drawing, chose the correct reading - but a computer cannot. Therefore it is important to know the rules that ContourCAM employs when reading a drawing. And it is these rules, and the commands that use these rules, we will describe in this section.

ContourCAM’s most important task is to generate NC code based on a drawing. The code generation is done by the commands Milling, Turning and Cutting depending of the type of machine tool you wish to program

5.4.1 Generating NC code from a drawing

In this section we discuss generation of NC code based on a drawing of toolpaths. The principle is to convert a drawing consisting of lines and arcs to the corresponding movements in an NC program. Additionally, there are options to regulate feed rate, tool change, radius compensation etc.

5.4.1.1 Drawing format

A drawing file used for NC generating consist of geometries and texts. Texts can either be single commands (these always start with a dollar sign, $) or they can be a part of a geometry. A geometry is a coherent set of the entities lines, arcs, arrows and texts. The idea is that each geometry is run through by using each entity exactly one time. If a whole geometry is not run through it is considered being more than one geometry. Entities of other types than the ones mentioned are ignored.

5.4.1.2 Types of entities

Noted below is a list of those drawing entities being used for NC generation and their meaning.

Arrow lines Indicates a linear movement with feed rate with the limitation that the CNC unit can only move in the direction of the arrowhead. Arrow lines are normally used to set the direction

Lines (type 1) Indicates a linear movement with feed rate.

Lines (type 5) Indicates rapid traverse.

Arcs Indicates a circular movement on the CNC unit.

Circles For full circles the direction of the circulation is noted by using the line type. Clockwise = 0 (full) and counter-clockwise = 1 (dashed).

5.4.1.3 Defining of toolpath

Before the preparation of a geometry can begin a start point must be selected. You can, by using start numbers (texts starting with a number), indicate start points for geometries. When the program must locate a new point of start the start point, that has not been used yet, with the lowest number is selected. When all start points has been used the preparation will continue on the closest stop point of an entity, if autotrace was selected, else the user will be asked for help.

In the following we will take a closer look at the rules for the preparation, i.e. in what order one would like this executed, indication of speed, depth of the cutting, radius compensation etc.

First we will look at how the toolpath through a geometry is decided. Each time we come to the stop point of an entity, all those entities, stop points of which are within the distance noted as tolerance in the editor are found. Then the entity with the highest priority is selected and the preparation continues.

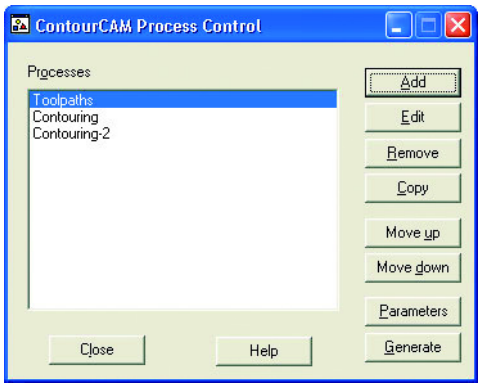
The priority of entities is as follows: Texts as always handled first. If the color is black (1. color column) the text will be interpreted (more about this later) otherwise the color number (color number in the color palette) is decreased by one and will eventually be interpreted later. The remaining entity types are given priorities according to their color: The lower the color number the higher priority. If two or more entities have the same color, they are given priorities in the following way:

- Arrow lines (preparation never starts at the arrowhead)
- Full circles
- Arcs
- Lines (type 0 (full))
- Lines (type 5 (dotted))

When a point is reached, from which there are no more entities, the next geometry is continued with until all geometries have been dealt with

5.4.2 Working processes

When you have finished the construction of the object, and you wish to determine, how your drawing must be prepared you choose ContourCAM from the main menu and from the pull down menu, if the item is to be milled the option Milling. When you choose the command the dialog below is displayed.



Add: Here you have to define what kind of preparation you wish for the current drawing. When the command is activated, you can choose among a number of preparation processes depending on whether you are working with milling, turning etc..

Edit: If you wish to edit a process you choose Edit. The dialog with the entered values is displayed on screen, and then the parameters can be changed.

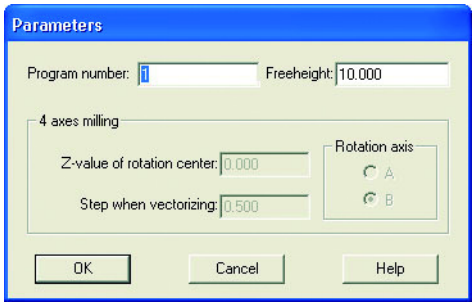
Remove: If a process is highlighted it will be removed if you press this button.

Copy: Appends a copy of the highlighted process. Please note that not all processes can be copied.

Move up/down: If you wish to change the order of the chosen processes you can move the highlighted one step up or down respectively.

Parameters: When you choose parameters a dialog for adjustment of the general parameters for the NC code is displayed, see Section 5.5.2.1

Generate: When you want to convert the defined processes to an NC program you press this button, see Section 1.5.2.2.



5.4.2.1 Parameters

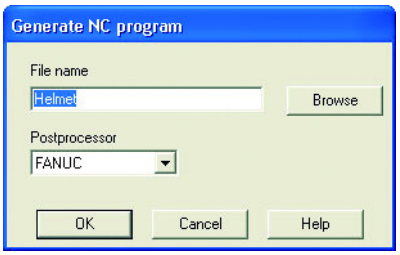
When you press the button Parameters in the process control the dialog below is displayed.

Program number: Enter the program number for the NC-program. This number will be inserted where the system variable {pno} is used, typically in the sequence for the program start.

Free height: Enter the Z value for the height where positioning movements in rapid traverse are allowed.

5.4.2.2 Generating of an NC program

When the command Generate is activated the dialog below is displayed.



File name: The file where the NC program is saved. Automatically a name corresponding to the name of the drawing is suggested. If you do not enter a certain path the file will be saved in the directory stated in the postprocessor or in the configuration of ContourCAM. If the file name has no extension the default extension will be added automatically.

Browse: Can be used if you wish to have an overlook of the files already existing (exit by <Esc>) or if you wish to overwrite an existing file (exit by <Enter>).

Postprocessor: Here you choose the postprocessor you wish to use. The default postprocesssor will be preselected the first time after the last one used.

5.5 Milling

In the following the preparation processes available for milling in 2 and 2½D will be described:

- Milling of a toolpath.
- Milling of a contour.
- Pocket milling.
- Boring etc.
- Engraving.

5.5.1 Milling of a toolpath

The toolpath process is used to directly convert the lines and arc of the drawing to movements in the NC program. This is the process that comes closest to directly programming on the control, in that you have to draw each horizontal movement you want the milling machine to perform, most of the vertical movement are generated automatically.

Operation starts at each start number by descending to working level then the geometry is followed using the standard rules for path finding described earlier. You may place text command at entity end points which will influence the generation of the NC-code. The texts will be handled when they are met during the preparation of geometries. More commands can be collected into one text separated by commas or space, you can also state commands in texts that defines start numbers. The following commands are available:

Z=d Fixes the cutting depth to d. Is effective until something else is stated or to the end of the geometry. If the command is not placed at the beginning of the geometry. The change of height will take place during the movement to the point. The command has no effect, if the type of the machine is fixed for anything but a milling machine.

ZD=d Fixes the cutting depth d. The difference between this and the command above is that the change of height always will be carried out by a vertical movement.

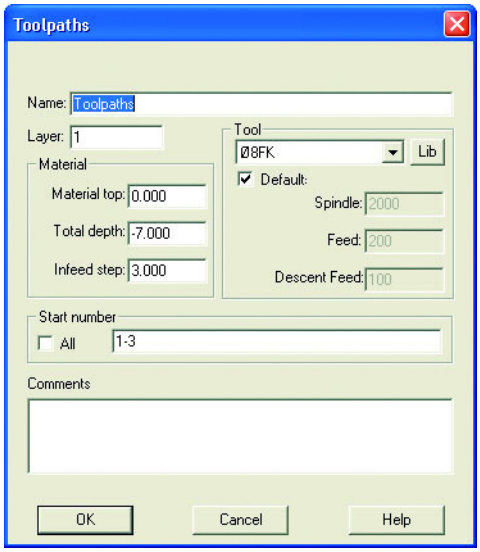
RL Turns on left sided tool compensation in the subsequent movement.

RR Turns on right sided tool compensation in the subsequent movement.

R0 Turns down tool compensation in the subsequent movement.

F=d States the feed rate to d. Is active until anything else is stated or until the end of the geometry.

Below you se the dialog for milling of a toolpath:



Name: Enter the name under which this process should be recognized.

Layer: Enter the number of the layer where the elements of the drawing which are to be used for this process are drawn.

Material top: Enter the Z-value of the top of the material in which the path are to be milled.

Total depth: Enter the Z-value for the button of the path.

Infeed step: Enter the maximum depth of material to be removed in each pass.

Tool: From the list of tools you choose the tool, which is to be used for the actual preparation.

Lib: Pressing this button brings up the tool library enabling you to define new tools etc.

Defaults: If this option is checked the fields spindle, feed and descent feed are filled out with the default values.

Spindle: Enter the speed of the spindle.

Feed: Enter the feed wanted. It is possible to preselect the speed of feed and spindle when you define your tool. If a tool is defined with preselected feed/ spindle these values will be adjusted automatically when the tool is selected.

Descent feed: Enter feedrate for the first vertical movement to the object.

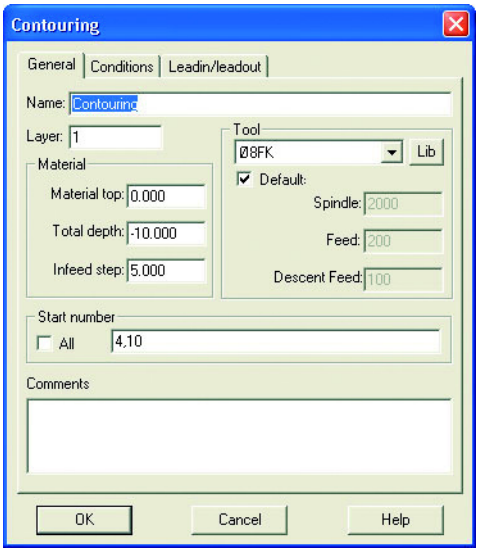
Start numbers: Enter the start numbers for which the stated values shall apply. You can list the individual numbers separated by commas, or write intervals using a hyphen, e.g. “1-5,7,9” means the numbers 1,2,3,4,5,7 and 9. If All is checked all start numbers in the entered layer will be used.

Comments: If you wish to add comments to the actual process you can do it here. The comment will be written in the NC program marked with apostrophes (’) in front of the text.

5.5.2 Milling of contours

The contouring process is used to mill one side of a contour. You can do contouring on either open or closed geometries. For open geometries you must specify which side to mill. For closed geometries ContourCAM determines what is the internal and external sides. Thus it suffices to specify, if you want to mill the inside or outside of the geometries and if you want to use climb or conventional milling. ContourCAM automatically generates lead ins and lead outs so you should not draw these.

Now we are to look at the dialog for contouring. The dialog for general adjustments is identical to the dialog for toolpath milling, therefore we will only describe the tab Conditions where information special to contouring is entered. The dialog is shown below:



Closed geometries: By clicking in the appropriate radio button, you decide, whether the preparation of the closed contours should be performed using climb or conventional milling and also if the preparation should be carried out on the internal or external side or the contours. If the process only applies to open contours this last field should be ignored.

Open geometries: Here you decide, if the preparation of open contours should be carried out on the right or left hand side of the contours. If the process only applies to closed contours this field should be ignored.

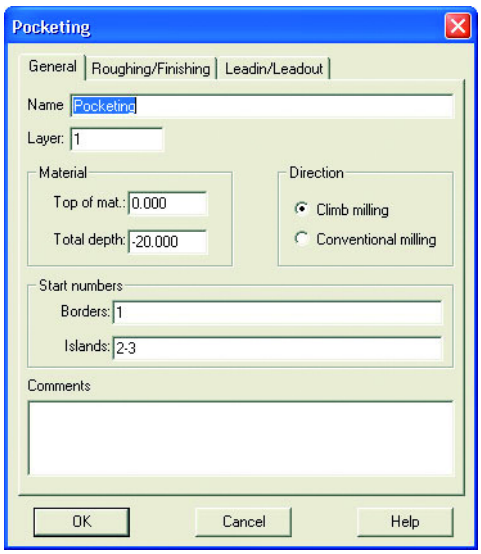
Radius compensation: Here you decide if the tool compensation of the contours should be made by your control, or if ContourCAM must compensate for the tool, before the NC code is generated. It is often preferable to leave the tool compensation to the control (if the control is able to) as it is much easier to adjust the compensation on the control, if the tool does not correspond to the measurements exactly.

Lead In/Out: Here it is possible to select, if you want a circular or a linear lead in and lead out as start/ending of the preparation. If you choose circular lead in/out you must enter radius and angle and if you choose linear lead in/out you must enter the length of the line and the angle wanted.

Overshoot: Here you can enter a value that will determine the distance between the lead in and the lead out to avoid that material is left.

5.5.3 Pocket milling

Pocket milling is used to remove material in an enclosed area. You can specify any number of borders and islands, the only restrictions are that all geometries that are borders or islands must be closed, i.e. without start and end, and each island must be completely contained within a border. ContourCAM will generate toolpaths with minimum rapid traverse. When you choose the command pocket milling the dialog below is displayed.



Name: Enter the name under which the information of the actual pocket milling should be saved. The name is transferred to the process control from where you select it if for instance you want to edit the information later.

Layer: Enter the number of the layer where your object is drawn.

Top of material: Here you enter the Z value for the top of the material.

Total depth: Enter the Z value of the bottom of the pocket.

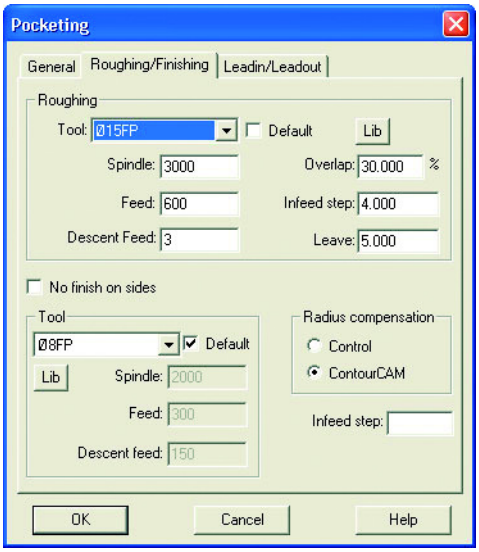
Direction: Click the mouse button in one of the round fields to decide if the pocket milling shall be performed using climb or conventional milling.

Start numbers, Borders: Enter the numbers of the closed contours surrounding the areas to be milled.

Start numbers, Islands: Here you enter the numbers of the island(s) which must be left when the pocket milling is finished. An island has to be placed inside the border of the area to be pocket milled.

Comments: If you want you can attach comments to the task.

Now we will look at the dialog appearing, when you click **Roughing/Finishing**. The dialog is shown below:



Tool: From your list of tools you choose the tool you wish to use for the roughing of the pocket.

Spindle: Enter the speed of the spindle (if preselected it works as under feed).

Feed: Enter the feed wanted (if you choose a tool with preselected values those values will be added automatically).

Descent feed: The feed rate used when descenting into the material.

Overlap: Enter the value in percent for the overlap you wish for each lap.

Infeed step: Enter the maximum depth of material to be removed in each pass.

Leave: Enter how much material to leave on the sides for final cutting.

No finish on sides: If you do not want to generate code for final cutting you just click this field.

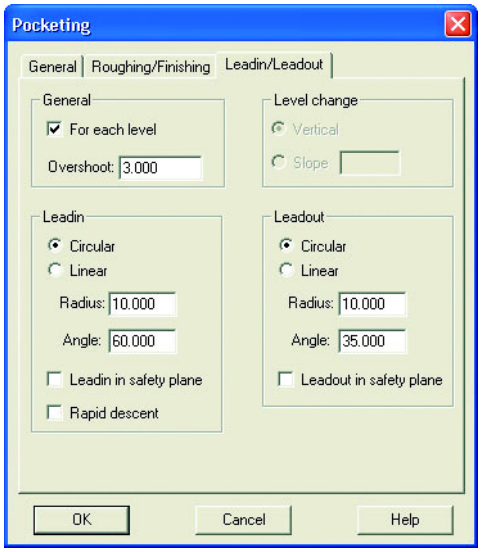
Tool: From your list of tools you choose the tool you wish to use for the finishing of the pocket.

Spindle: Enter the speed of the spindle (if preselected it works as under feed).

Feed: Enter the feed wanted (if you choose a tool with preselected values those values will be added automatically).

Descent feed: The feed rate used when descenting into the material.

Radius compensation: Here you decide if the tool compensation of the contours should be made by your controller or if ContourCAM must compensate for the tool before the NC code is generated. It is often preferable to leave the tool compensation to the control (if the control is able to) as it is much easier to adjust the compensation on the control if the tool does not correspond to the measurements exactly.



Lead In/Out: Here it is possible to choose if you want a circular or a linear lead in respectively lead out as entrance/ending of the preparation. If you choose circular lead in/out you must enter radius and angle and if you choose linear lead in/out you must enter the length of the line and the angle wanted.

Overshoot: Here you can enter a value that will determine the distance between the lead in and the lead out to avoid that material is left.

5.5.4 Boring etc.

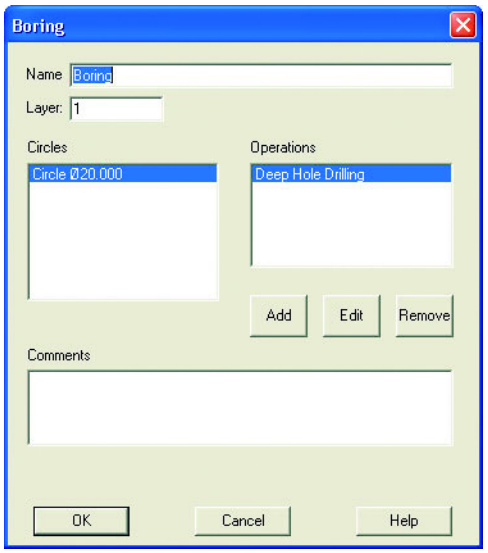
The boring process enables you to call canned machine cycles for boring, drilling, tapping etc. on the circles in the drawing. When this process activated ContourCAM will search the drawing for full circles in the indicated layer. The circles are grouped according to their diameters and the different sizes will appear in the list of circles. For each group of circles you can add up to ten operations. When generating NC code ContourCAM will optimize for minimum travel and tool change. When you choose boring from the menu the dialog below is displayed.

Name: Enter the name under which the information of the actual preparation should be saved. The name is transferred to the dialog for boring from where you can select it later if you wish to edit some parameters.

Layer: Enter the number of the layer, where the object is drawn.

Circles: ContourCAM automatically detects the diameters of circles in your drawing and classifies them. It helps you to select the most sensible tools for the making of each hole.

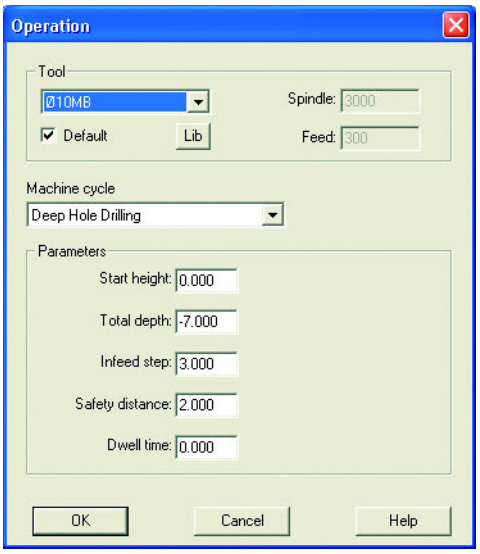
Operations: When one diameter measurement is highlighted this field shows, which operations you have selected precisely for those holes.



Edit: If you wish to edit a selected cycle the cycle in question must be highlighted before clicking edit.

Delete: Deletes the highlighted cycle.

Comment: If you want enter comments to the task here.



When you click Add from the dialog another dialog appears.

Tool: From your list of tools you choose the tool you wish to use for the actual job.

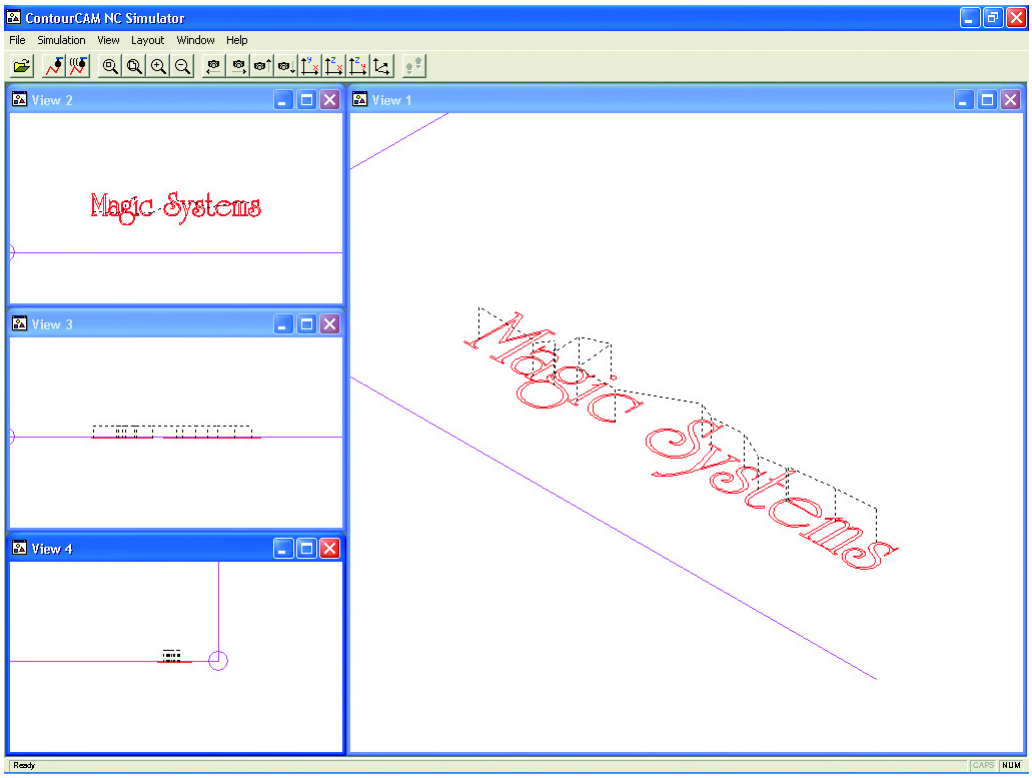
Feed: Enter the feed wanted (if you choose a tool with preselected values those values will be added automatically).

Spindle: Enter the speed of the spindle (if preselected it works as under feed).

Machine cycle: From the library of cycles you choose the cycle you wish to use. If you click the field to the right , a complete list showing the cycles available appears

Parameters: Depending on the cycle chosen the relevant parameters for carrying out the cycle appears. You enter the values wanted then you click the OK button to accept the values entered.

5.5.5 Engraving



With the engraving process you can generate NC code for engraving any drawing on a flat surface. ContourCAM will generate NC movements for all lines and arc in the indicated layer. The code will be optimized for minimum rapid traverse. The dialog is shown below:

Name: Enter the name under which the this process shall be recognized.

Layer: Enter the number of the layer where your object is drawn.

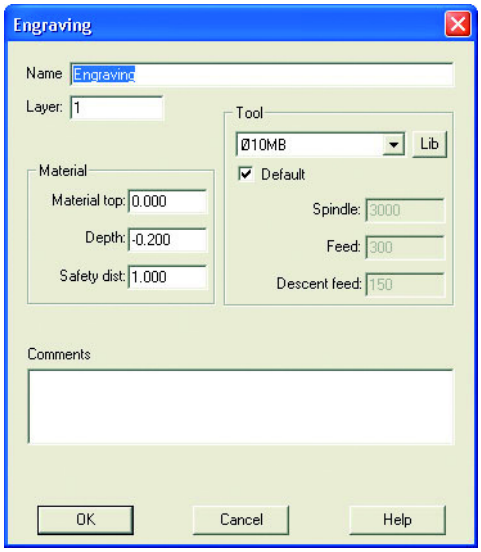
Top of material: Here you enter the Z value for the top of the material.

Depth: Enter the Z value for the distance between the top of material and the button of the engraving.

Safety: Enter the relative distance to the material where movements in rapid traverse is allowed.

Tool: From your list of tools you choose the tool you wish to use for the actual job.

Spindle: Enter the speed of the spindle (if preselected it works as under feed).



Feed: Enter the feed wanted (if you choose a tool with preselected values those values will be added automatically).

Descent feed: Enter feed rate for the first vertical movement to the object.

Comment: If you want you can attach comments to the task.

5.6 Cutting

In the following the preparation processes available for cutting will be described:

- Cutting paths.
- Contour cutting.

5.6.1 Cutting paths

The cutting paths process is used to directly convert the lines and arcs of the drawing to movements in the NC program. This is the process that comes closest to directly programming on the control, in that you have to draw each horizontal movement you want the cutting machine to perform.

Operation starts at each start number by lighting the torch, then the geometry is followed using the standard rules for path finding described earlier. You may place text command at entity end points, which will influence the generation of the NC-code. The texts will be handled when they are met during the preparation

of geometries. More commands can be collected into one text separated by commas or space, you can also state commands in texts that defines start numbers. The following commands are available:

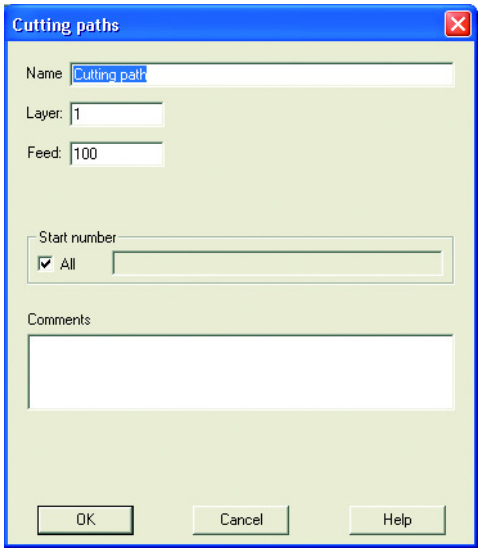
RL Turns on left sided tool compensation in the subsequent movement.

RR Turns on right sided tool compensation in the subsequent movement.

R0 Turns down tool compensation in the subsequent movement.

F=d States the feed rate to d. Is active until anything else is stated or until the end of the geometry.

Below you se the dialog for cutting paths:



Name: Enter the name under which this process should be recognized.

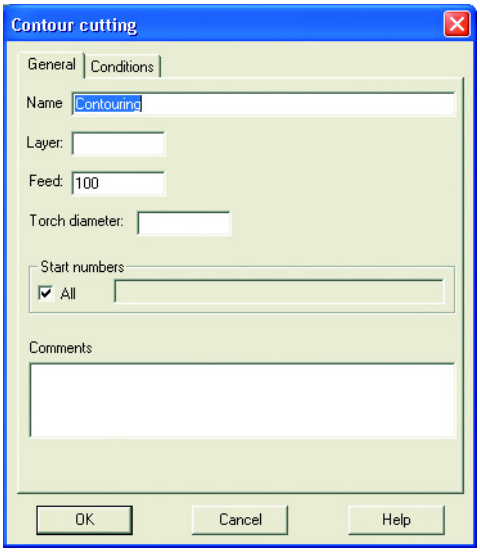
Layer: Enter the number of the layer where the elements of the drawing which are to be used for this process, are drawn.

Feed: Enter the feed rate. You may leave this empty, if the postprocessor does not generate codes for feed rates.

Start numbers: Enter the start numbers for which the stated values shall apply. If All is checked all start numbers in the entered layer will be used.

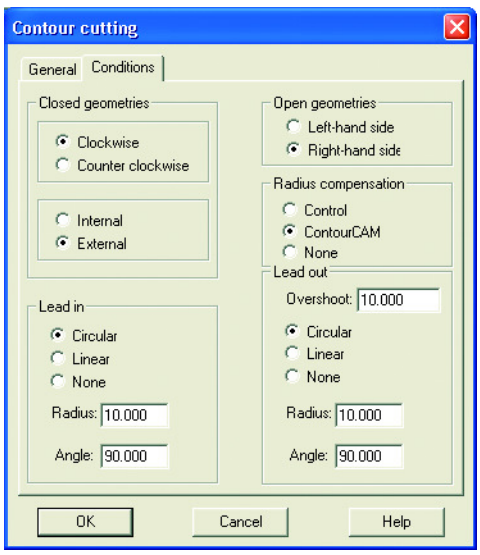
Comments: If you wish to add comments to the actual process you can do it here. The comment will be written in the NC program marked with apostrophes (') in front of the text.

5.6.2 Contour cutting



The contouring process is used to cut one side of a contour with compensation for torch diameter. You can do contouring on either open or closed geometries. For open geometries you must specify which side to cut. For closed geometries ContourCAM determines, which are the internal and which are the external sides. Thus it suffices for you to specify, if you want to cut the inside or outside of the geometries and if you want to cut clockwise or counter-clockwise. ContourCAM automatically generates lead ins and lead outs so you should not draw these.

Now we take a look at the dialog for contouring. The dialog for general adjustments is almost identical to the dialog for cutting paths, the only difference being a field for entering the torch diameter. Therefore we will only describe the tab Conditions where information special to contouring is entered. The dialog is shown below:



Closed geometries: By clicking the appropriate radio button it is decided, whether the preparation of the closed contours should be performed clockwise or counter-clockwise and also if the cutting should be carried out on the internal or external side or the contours. If the process only applies to open contours this field should be ignored.

Open geometries: Here you decide if the preparation of open contours should be carried out on the right or left hand side of the contours. If the process only applies to closed contours this field should be ignored.

Radius compensation: Here you decide, if the torch radius compensation of the contours should be made by your control, or if ContourCAM should compensate for the torch before the NC code is generated. It is often preferable to leave the torch compensation

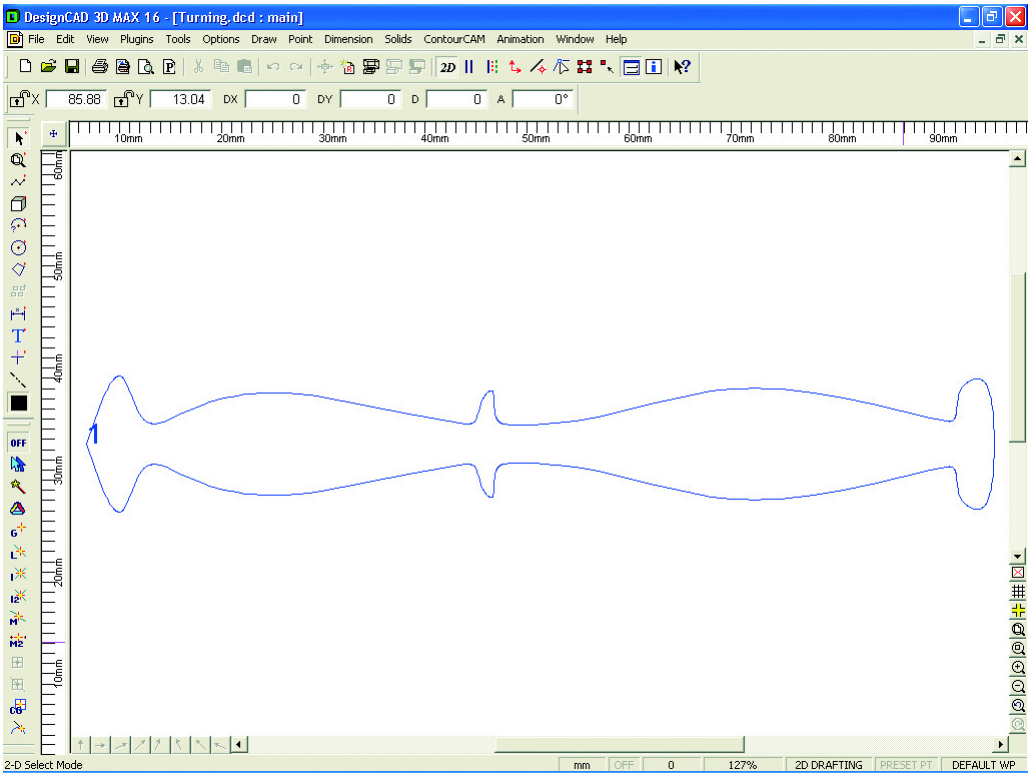
to the control (if the control is able to), as it is much easier to adjust the compensation on the control, if the torch does not correspond to the measurements exactly.

Lead In/Out: Here it is possible to choose, if you want a circular or a linear lead in respectively lead out as start/ending of the preparation. If you choose circular lead in/out you must enter radius and angle, and if you choose linear lead in/out you must enter the length of the line and the angle wanted.

Overshoot: Here you can enter a value that will determine the distance between the lead in and the lead out to avoid that material is left.

5.7 Turning

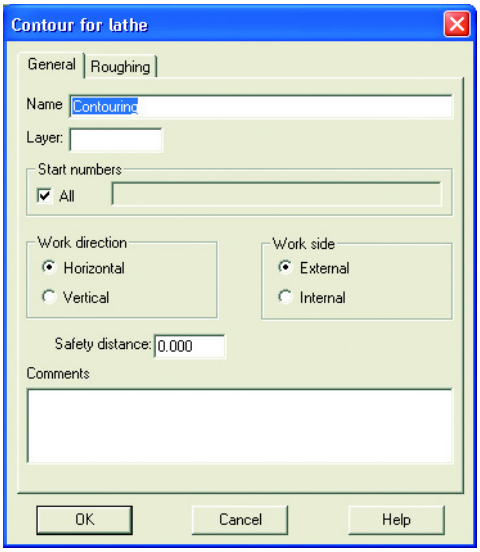
5.7.1 Contour turning


DesignCAD 3D MAX 16 - [Turning.dcd : main]
File Edit View Plugins Tools Options Draw Point Dimension Solids ContourCAM Animation Window Help
X 85.88 Y 13.04 DX 0 DY 0 D 0 A 0°
10mm 20mm 30mm 40mm 50mm 60mm 70mm 80mm 90mm
1
2-D Select Mode
mm OFF 0 127% 2D DRAFTING PRESET PT DEFAULT WP

In the process contour turning it is possible to generate both roughing and finishing. Most new lathes have machine cycles to do the roughing, but in ContourCAM we have added a command to generate roughing for owners of older lathes.

To use it you have to draw the final contour, vectorize it and add a start number at the end of the object, where the work must start.

From the ContourCAM menu you select Turning and Add and you get the following dialogs:



Name: Here you enter the name of the process. You can have more than one process in and NC-program.

Layer: Here you enter the layer number in which are drawn the elements of the drawing to be used in this process.

Start numbers: Enter the start numbers for which the stated values shall apply. If All is checked all start numbers in the entered layer will be used.

Work direction: Here you set a dot in either horizontal or vertical.

Work side: Here you set a dot in either external or internal.

Safety distance: Here you set the safety distance.

Comments: Her you can place any comments you might have.

ContourCAM 3D 6

The 3D parts of many CAD/CAM systems are constructed such that the user draws the objects directly in 3D. But as working directly in 3D requires more abilities than working in 2D, we have chosen another strategy for ContourCAM.

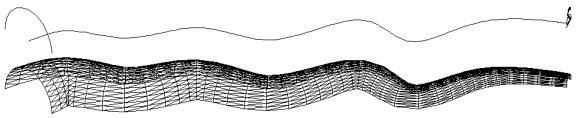
We have, however, introduced a function, which we call **Carpet**, whereby you can mill actual 3D surfaces, but so far it does not work on all types.

The 3D part of ContourCAM is based on solving as many task as possible using 2D drawings only. In ContourCAM there is generally only one way of defining 3D surfaces; we call this principal **Snake** programming.

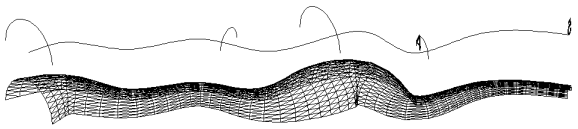
ContourCAM's **Snake** programming technique offers the following:

- Generation of finishing programs for 3D surfaces.
- Generation of roughing programs for 3D surfaces.
- The NC programs can be radius compensated for every thing from end to ball cutters with check for undercutting.
- Engraving in 3D surfaces.
- Generation of DesignCAD 3D drawings showing, what the surface will look.like

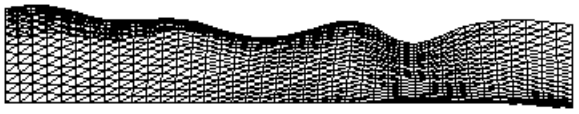
A snake is described by one or two transmission profiles and a number of shift profiles. If it is possible to describe each profile in one plane only, you can draw them using DesignCAD 2D; if not the profiles must be drawn in DesignCAD 3D. Once the profiles are constructed and you have told what is transmission and what are shift profiles, the snake program is started and ContourCAM will generate the surface.



Before we start describing the details of the snakes you must gain an understanding of what transmission and shift profiles are. If we imagine that we were to make a program of a snake (the animal that is) the transmission profile would be the spine of the snake. The shift profiles would be cross sections at the head and tail as shown in the above illustration.



If the snake had just eaten it would bulge in the middle. In this case it would not be enough to describe the snake using two shift profile - 4 or more are required to describe such a shape, as shown in the above illustration. It is possible to use more than 2 shift profiles; only the amount of available memory limits the number of shift profiles.



Furthermore, it is not always possible to describe a surface using only one transmission profile. Shown here is a surface that cannot be described using only one transmission profile as the shifts vary in width. Using ContourCAM it is possible to describe surfaces using two transmission profiles and several shift profiles.

When ContourCAM creates a surface based on transmission and shift profiles it first divides each profile into a number of short line segments. All shift profiles must be divided into the same number of line segments. Likewise, if there are more transmission profiles they must be divided into an equal number of line segments. ContourCAM then forms a net or grid by letting the shift profiles “move” down then transmission profile(s). When moving from one shift to the next the shape is gradually changed from that of the former to that of the latter.

Below we describe the theory and by examples show how the 2 snakes are used. After reading the theory we suggest that you try out the tutorial before starting on your own.

6.1 Snake programming

6.1.1 Snake-1 programming from DesignCAD 2D

We start by describing how you program snakes in DesignCAD 2D, and later we will discuss snake programming in DesignCAD 3D.

The first to do when construction a snake is to draw the profiles. You determine what is to be the transmission profile and the shift profiles. Using DesignCAD's drawing facilities you draw the profiles; all profiles must be in the same drawing and reside in the same layer.

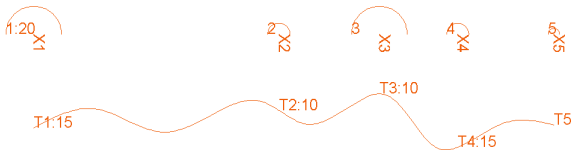
When the profiles are constructed you specify - using text - which are transmission and shift profiles. For each shift profile there must be a text on the transmission profile(s) specifying what shift is to be attached at this point and how many line pieces the profile should be divided into until the next specification.

When using the snake modules the following rules must be observed:

- The profiles may only consists of lines and arcs. If a profile drawn using e.g. a curve this must be vectorized before the snake can be generated.
- On each profile text are attached at the start point of the geometry. The text contains information on what profile it is, and how many pieces the profile should be divided into.
- It is important that texts are attached to the endpoint of an entity otherwise ContourCAM will not be able to make the connection between the text and the entity. If you want to attach a text to an internal point of a polyline you must break the polyline at this point.

6.1.2 Basic snake programming

When programming snakes in DesignCAD 2D both transmission and shift profiles are drawn in the same plane. To decide the position of the surface ContourCAM uses certain rules when it reads a drawing as a snake.



In this illustration a snake with one transmission profile drawn in DesignCAD 2D is shown. When ContourCAM reads this drawing the following happens:

A T followed by a number means that the geometry in question is the transmission profile, the number indicates that the shift profile with that number is to be attached at this location.

ContourCAM understands the transmission profile as lying in the XY plane, while the shift profiles are raised to vertical and are placed at their attach points on the transmission profile.

6.1.2.1 The shift profiles

A shift profile is described by the text code **n[:m]** (where **n** and **m** are numbers and brackets **[ ]** indicate optional text). For example "**1:20**", "**2**".

n: Indicates the number of the shift profile in question.

m: Indicates how many line pieces the shift profile is to be divided into, from this text and forth. For the first shift the text must include :m for the rest it is not necessary.

By default it is the start point (i.e. the point where the text is attached) that is attach to the first transmission profile and if there are two transmission profiles, the end point that is attach to the second transmission profile. To change the default behaviour the text code Xn can be used to specify the point which corresponds to the attach point on the first transmission profile. This text code need not be connected to the shift profile in any other way than having the same number. Likewise, you can use the text code Yn the change where the shift profile is attached to the second transmission profile.

It is important to remember that a shift profile must never be a closed geometry.

If more shift profiles have the same shape you only have to draw one. Let us say that the shifts 1,3 and 5 have one shape while shifts 2 and 4 have another, and say we divide the shift profiles into 50 line segments. We would then draw the two shapes, and to the first attach the text “1,3,5:50” and to the second the text “2,4”. Observe we only indicate the division on the first shape. Also note that if you use the text code X1 it will apply for all of the shifts 1, 3 and 5.

6.1.2.2 The transmission profiles

The general form of the texts of the (first) transmission profile is **Tn[:m]** (brackets **[ ]** indicate optional text).

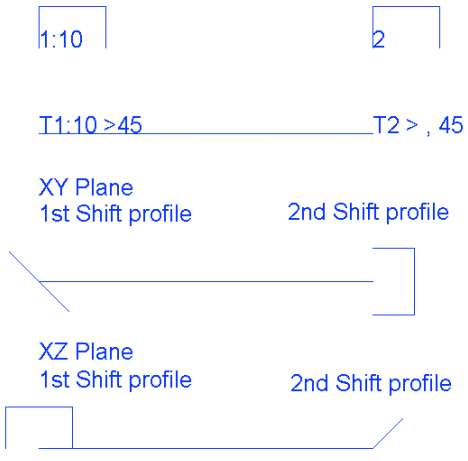
n: Indicates which shift profile is to be attached here.

m: Indicates how many line segments the transmission profile is to be divided into from this text to the next; we call knots the endpoints of these line segments. For each shift profile attachment m must be specified except for the last of an open transmission profile.

Examples of texts on the transmission profile: "**T1:10**", "**T3**".

The shift profiles must be enumerated by increasing numbers. There are no further demands on the numbers, like with which to start and steps between numbers. If the transmission profile is a closed geometry, the entity that starts the profile should have a color of lower priority than the entity which ends the profile. If for example the transmission profile is black (priority 1) the entity which ends the profile could be made blue (priority 3). If several shift profiles are to be attached at the same point on the transmission profile the directive may be written separated by commas, e.g. “**T1:10, T2:10**”.

As mentioned earlier a transmission profile drawn in DesignCAD will by default be interpreted as lying in the XY plane, while the shift profiles are placed vertically. The shift profiles can either be rotated such that their planes are perpendicular to the tangents of the transmission profile at the attach points, or be placed with their plane parallel with the X axis.



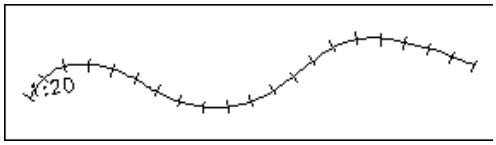
If one or more of the profiles are not to be placed perpendicular to the transmission plane or tangent, rotation directives can be specified at the attach point. The code to use for specifying attach points in such cases is of the following form: **Tn[:m][>[d][,e]]**. For example: "**T1:10>20**", "**T3>,30**", "**T5>10,15**". **d** indicates the rotation in degrees around the normal vector of the plane of the transmission plane, i.e. deviation from the tangent. **e** indicates the rotation in degrees of the shift relative to the plane of the transmission plane, i.e. deviation from vertical.

This illustration is an example of how the rotation directives works.

6.1.3 The placements and limitations of the knots

It is important to have an understanding of how ContourCAM calculates the surface and hence the tool paths. The starting point is the number of line segments into which you have specified for ContourCAM to divide the profiles. For example if you specify **1:10** shift 1 is divided into 10 line segments which equals 11 knots on the profile.

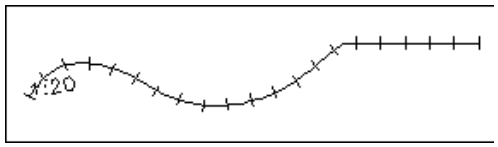
Knots on a smooth curve



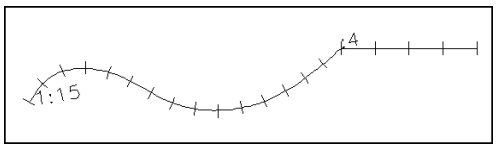
Here you can obtain an understanding of the situations, in which you run into troubles.

Normally, if the shifts are curve movements, as in this illustration, there are no problems.

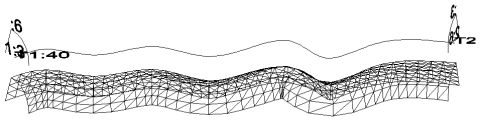
A curve with an edge



In this illustration there could be problems since the profile has an edge. If the distance from the start point to the edge is not precise the edge will disappear from the surface.



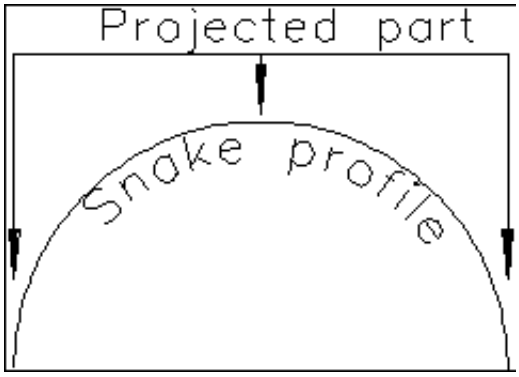
It might be vital to have such an edge. In those cases it is necessary to be more specific about how to divide the profile, as shown in this illustration. Furthermore, one needs to consider the shape of the adjacent shift profiles, more about this later.



To minimize the size of NC program it is a good idea to decide how you wish the NC program generated (along the transmission profile or the shift profiles) and how close the toolpaths should be. For instance if a distance of 1 mm between the toolpaths is desired and they should follow the transmission profile, it is the points on the shift profiles that decide the density of the toolpaths. Thus it is preferable to have a distance of 1 mm between the knots on the shift profiles. If the transmission profile is a

smooth curve the distance between its knots might be more than 1 mm. These optimizations of the NC program is of course on the expense of a deviation from the true profile. It is thus necessary to consider, if it is an acceptable deviation. In the tutorial this will be exemplified.

6.1.4 Engraving in a snake



By projecting we understand that a given XY coordinate in the object you want to bend gets the Z value of the corresponding point in the snake.

In addition to making the tool-paths for milling the surface described by the snake, you can also engrave drawn objects in the surface. This is done by projecting the drawn elements onto the surface defined by the snake, i.e. for each point (X,Y) of the drawing corresponding Z value of the surface is used (with a given offset added).

The snake commands always does projection. This is because bending with true length is only possible in single curved surfaces.

The object to be bend should be in a layer for itself in the same drawing as the snake.

Bending is especially useful for putting text onto a 3D surface for engraving.

6.1.5 Creating a snake process

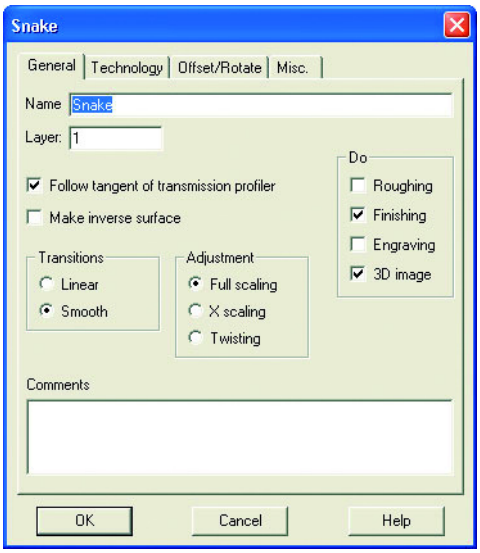
Snake processes are added to an NC program much the same way as pocket milling, contouring processes etc. You activate the process control for milling from then menu **ContourCAM=>Milling** in DesignCAD. Then press the Add button and select Snake.

Name: Enter the name of the operation to be carried out.

Layer: Enter the number of the layer where the actual drawing is saved.

Follow tangent of transmission profile: If this option is checked it means that the cut profiles regardless of the form of the transmission profile always must be placed perpendicular in proportion to it. If the option is unchecked the cuts must follow the turning of the transmission profile.

Make inverse surface: Choose this option if you are to produce for instance a mold consisting of a male and female part. The opposite of the one you programmed will be generated if you select this option.



Transition:

Linear: If you choose to connect the cuts by lines the movements between each cut will be linear which means that you will have an edged object.

Smooth: If you want a smooth transmission between the cuts you activate this option.

Adjustments:

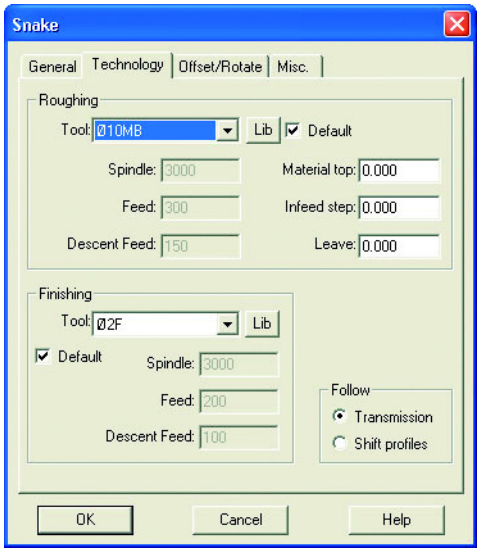
Full scaling: By full scaling the cut profiles will be scaled in XYZ to fit between the two transmission profiles. Example: if a semi circle is bigger than the distance between the two transmission profiles the form of the object will still be a semi circle.

X-scaling: By x-scaling it is understood that the cut profile does not change the y- and z-values during the scaling. Example: If the cut profile is a semi circle bigger than the distance between the two transmission profiles the cut will change to an ellipse with the same max. radius as the radius of the original semi circle.

Twisting: By twisting it is understood that the sides of the cut profiles will be perpendicular to the transmission profiles.

Do: The processes which you activate will be generated.

Comments: Under Comments you can add information to the general adjustments. When you filled out the dialog with the general information you choose the flag Technology and the dialog in the figure is displayed.



Roughing:

Tool: From the list of tools you choose the tool which is to be used for roughing.

Spindle: Enter the speed of the spindle wanted.

Feed: Enter speed of feed.

Decent feed: Enter the feed for vertical movement into the material.

Top of material: The Z-value for the top of the raw material.

Infeed step: Depth per each movement in Z.

Leave: The thickness of material you may want to leave for the final cut.

Finishing:

Tool: From the list of tools you choose the tool you wish to use for the final cut.

Spindle: Enter the speed of the spindle.

Feed: Enter the speed of feed.

Decent feed: Feed by vertical movements in the material.

NC-paths follow:

Transmission: If you want the milling carried out following the transmission profile you activate this option.

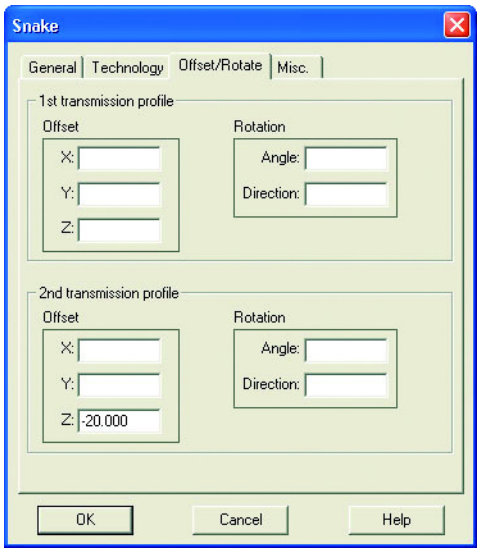
Shift: If the preparation must follow the shift profiles you activate this option.

If your transmission profile is placed otherwise than it is drawn it is possible to rotate or offset it. If that is necessary you click the flag Offset/Rotate, and the dialog in the figure is displayed.

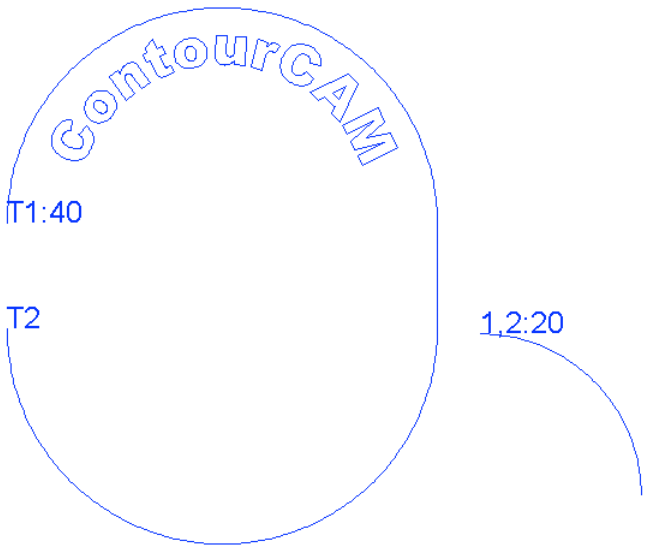
1st transmission profile | Offset: Here you can enter an offset of the first transmission profile. Often it will be in Z a value is entered as offset in X and Y can be obtained by moving the transmission profile in the drawing.

1st transmission profile | Rotation: If the transmission profile is not to lie in the XY plane, you can raise it to a given angle.

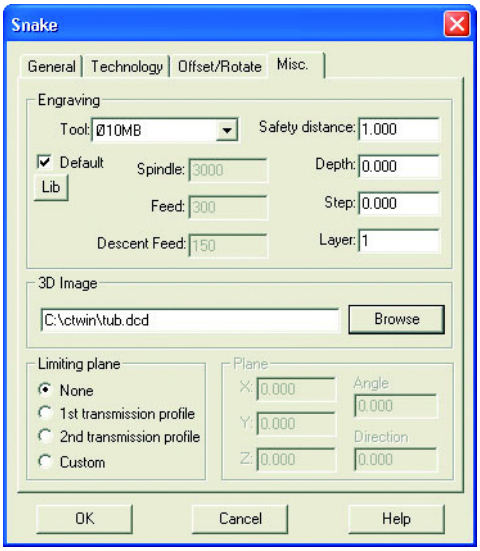
2nd transmission profile | Offset: Here an offset of the 2nd transmission profile can be entered. In addition to altering the Z value this feature is useful, if the first and second transmission profiles are connected. In this case they must be drawn some distance apart, and then the offset is used to reverse this distance.



2nd transmission profile | Rotation: If the transmission profile is not to lie in the XY plane, you can raise it to a given angle.



In ContourCAM it is possible to engrave text as well as drawings in the surfaces created by the snakes. The text or the drawing must be placed on the transmission or shift profile, where you wish the engraving to be executed (study this example).



When the text/drawing has been placed as you want you select the flag **Engraving**. The dialog in the figure will appear.

Tool: From the list of tools you choose the tool you wish to use for the engraving.

Spindle: Enter the speed of the spindle.

Feed: Enter the speed of feed.

Decent feed: Feed by vertical movements in the material.

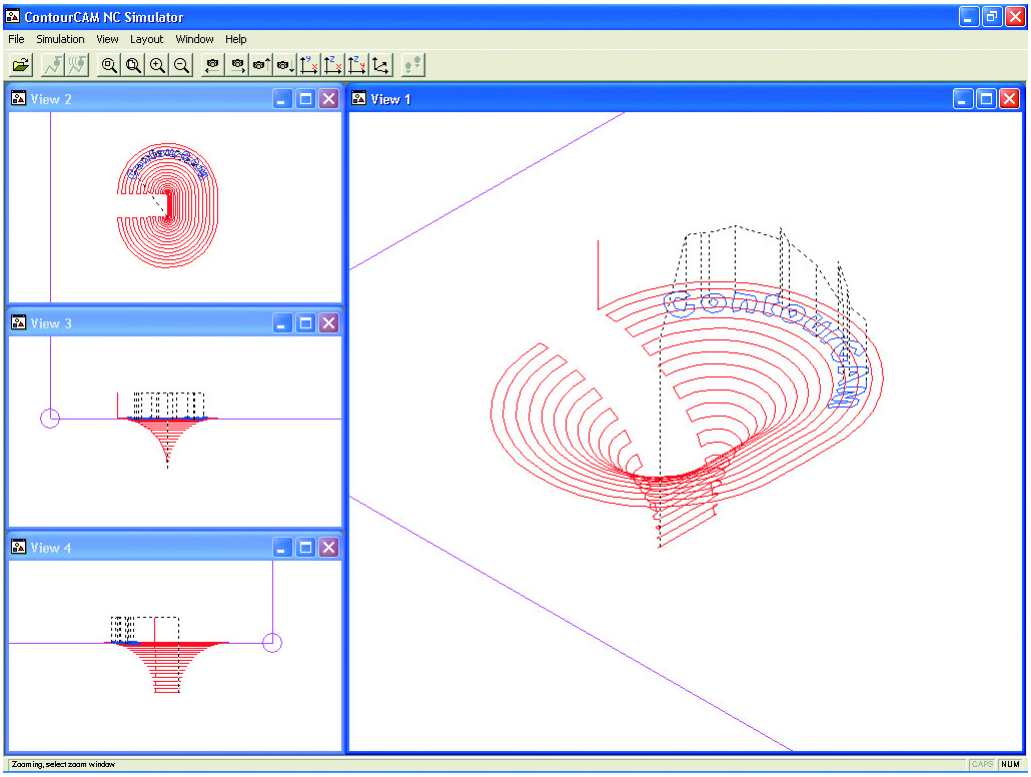
Safety distance: Enter the distance to the surface in mm where movements in rapid traverse are allowed.

Depth: Enter the depth into the surface to do the engraving.

Step: Enter the length of the linear movements. All movements of the drawing will be divided into line segments of this length to be able to follow the surface.

Layer: Enter the number of the layer where the text/drawing to be engraved into the surface is drawn.

3D Image: Here you enter the file name for the 3D image of the snake. If you check the make 3D image option on page one, ContourCAM will generate a DesignCAD 3D drawing of the surface defined by the snake. You can use this drawing to verify that the snake does indeed define the surface you intended. Furthermore, you can use it for shading to make a dashing model of the resulting item.

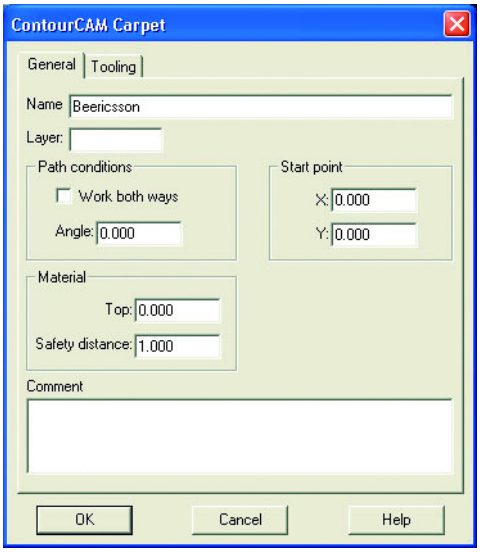


The picture above shows the engraving in the surface created by the snake system.

6.2 Milling Carpet

Menu : ContourCAM=>Milling=>Add=>Carpet

With the carpet process you can generate NC-code to finishing of an arbitrary 3D object. ContourCAM will generate parallel paths over the object, following the height of the drawing. Paths will only be generated over the drawing.



The general tab:

Name: Here you enter the name under which this process should be known.

Layer: The number of the layer in which your object is drawn is indicated here. If the field is empty all layers will be used.

Work both ways: Indicates if the milling machine should process back-and-forth, or travel back in rapid traverse.

Angle: Indicates angle of the paths to the X-axis.

Start point: Here you enter where the processing has start. It is not necessary to enter precise coordinates, ContourCAM will find the nearest starting point.

Material top: Here you enter the Z-value for the top of your material.

Safety distance: Here you enter the relative distance to material, to where z-movement in rapid traverse is allowed.

Comment: If you wish it, you can add a comment to the task here.

Tooling:

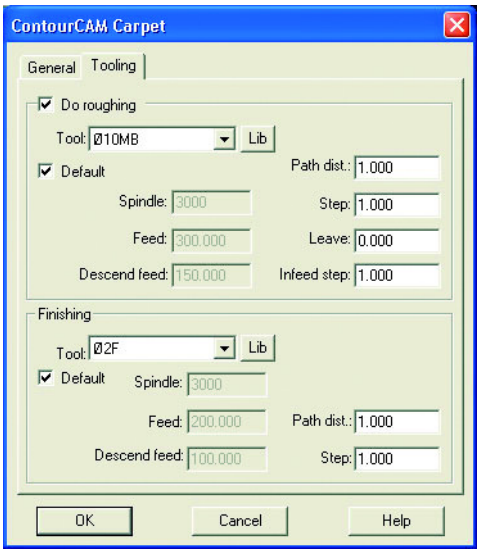
Do roughing: Here you indicate whether roughing should be done.

Tool: From your tool list choose the tools to be used for roughing and finishing respectively.

Default: If this is activated values for spindle and feed are inserted from the tool library, feed at submergence is calculated from the percentage of the feed rate given in options. When default is active the feed and spindle fields cannot be edited.

Lib: Clicking this button opens the tool library, so tool definitions can be edited. Note: Do not use this to of pick out tools.

Spindle: Here you enter the desired spindle speed. (if default, as for feed).



Feed: Here you enter the desired feed speed (if you choose a tool with default, the selected values for just this tool automatically be added).

Feed at submergence: Feed speed at vertical finishing in material.

Path distance: Here the distance between the parallel tool paths are indicated.

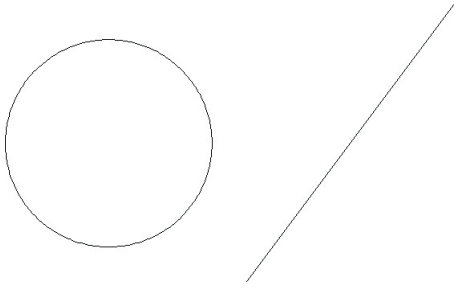
Tutorial ContourCAM 2D

7

In this chapter we present examples on the use of ContourCAM's command for NC programming.

7.1 Examples of drawing editing and nest commands

7.1.1 Fillet-2: Draw an arc between circle and line



Draw a circle through the following points, press <O>

<:> 30,30

<:> 45,30

Draw a line through the following points, press <V>

<:> 50,10

<:> 80,50

Now choose Edit from the main menu and from the pull down menu Trim/Fillet.

Now choose Fillet-2 from the menu.

1. Set a point close to the tangent point on the circle.

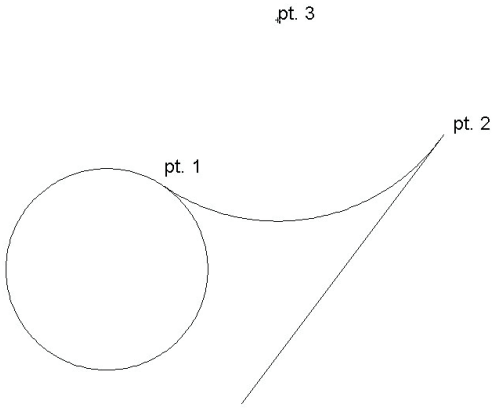
2. Set a point close to the tangent point on the line.

3. Set a point in the direction of the center of the rounding arc.

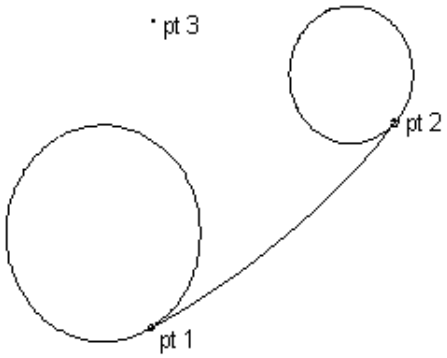
4. Enter radius of the rounding arc = 30.

Remember that the rotation direction is always counter-clockwise.

5. Clear the screen (Y)



7.1.2 Fillet-2: Draw an arc between 2 circles



1. Draw a circle with known center and a specified radius (circle 4). Radius 20 and center X30,Y30.

2. Draw a circle (circle 4) radius 15 and center X85,Y50.

Now we will make a rounding arc radius 200 at the bottom of the two circles. Choose Edit from the main menu and then Trim/Fillet from the pull down menu and then the command Fillet-2.

3. Set a point close to the tangent point on the circle to the left (the circle radius 20).

4. Set a point close to the tangent point on the circle to the right (circle radius 15).

5. Set a point in the direction of the center of the rounding arc.

6. Enter radius of rounding arc = 200.

Now we will make a fillet at the top of the two circles = radius 150.

Choose Edit from the main menu then Trim/Fillet from the pull down menu and then the command Fillet-2.

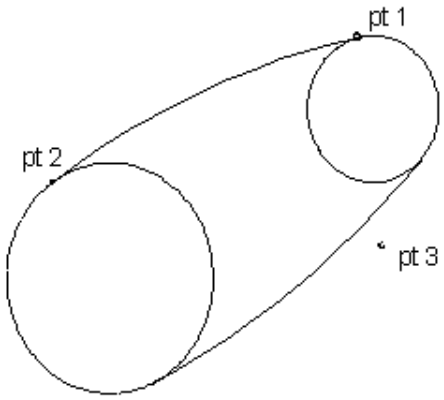
1. Set a point close to the tangent point on the circle (r15)

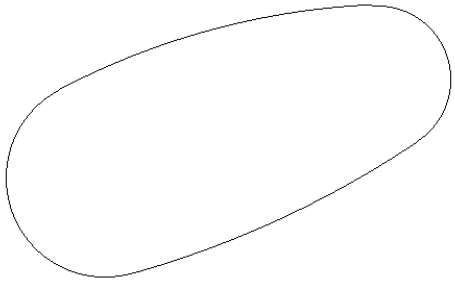
2. Set a point close to the tangent point on the circle (r20)

3. Set a point in the direction of the center of rounding arc

4. Enter radius 150.

The screen now shows the two circles connected by the fillets we just made.



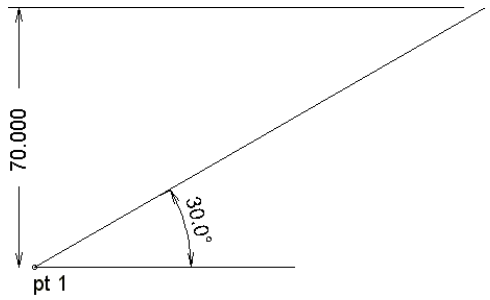


Now you can remove the parts of the circles inside the geometry. Use MagicTrim to cut it off automatically (see section 7.1.5).

7.1.3 Line PDXY: Drawing line dialog

Set a point in X30,Y30.

Choose Lines from the main menu and then Line PDXY from the pull down menu. The Line PDXY command allows you to describe a line in every possible way without forcing you to remember which point command to by used in every single case.



Answer the following questions and only key in the fields where the values are known:

1. P-angle = 30.

2. You do not know the distance.

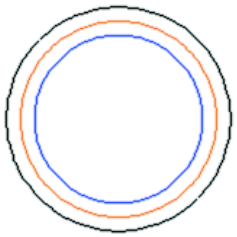
3. Relative X is not known.

4. Absolute X is not known.

5. Relative Y = 70.

6. Absolute Y is not known.

7.1.4 Delete all but specified color



1. Draw a circle radius 40 with center in X90,Y60.

2. Change color to red in the color table (<H>), Draw a circle radius 35 and the same center.

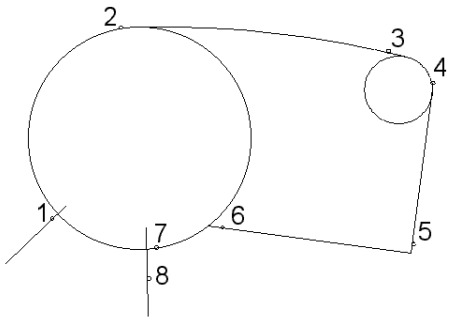
3. Change color to blue and draw a circle radius 30 and the same center.

4. Choose Edit and from there the command Delete but one color.

You are now asked to set a point on an entity with the color you wish to keep.

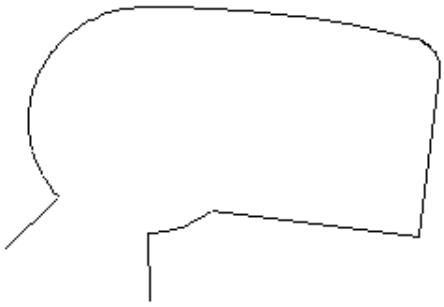
5. Set a point on the blue circle. The circles drawn in different colors are now removed automatically.

7.1.5 MagicTrim: Possibilities, Example 1



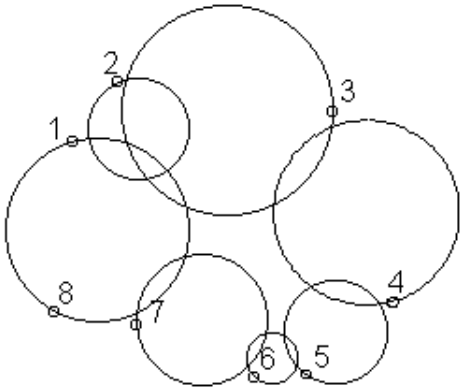
The command MagicTrim is activated from the menu Edit=>Trim/Fillet. MagicTrim is a general command for trimming where you point out the elements you want to be represented in your milling path for instance - MagicTrim automatically removes those part elements which are not part of the final item.

When activating the command you are to set points on the part elements you wish to be represented in the final preparation path. You are to set a point close to the intersection point with the following entity inside that part of the entity which is not to be cut off. In the example you can see succession of the entities and the position of the points. When you have set a point on all the entities you wish to keep press <Enter>.



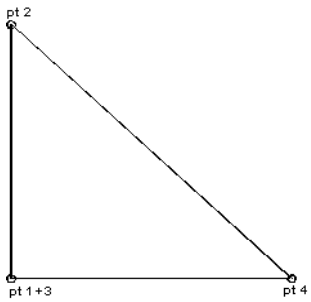
7.1.6 MagicTrim: Possibilities, Example 2

This example with circles only is as born to MagicTrim. It is possible to use the same geometry twice or more with the MagicTrim command.



It is always important to set the points close to the intersection point with the following geometry to indicate which intersection is in question in case there are more than one.

7.1.7 Nest-1: Nest an item to itself



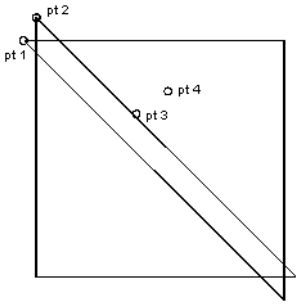
1. Draw a line through X30,Y30 and X30,Y100 and then a line through X30,Y30 and X100,Y30.

2. Snap to the endpoints and press (V) for Line.

3. You now have an triangle which you want to rotate 180 degrees to nest it to itself.

4. Select the triangle Edit/Entity select or the arrow in the upper left corner of the tool box.

5. Choose from the menu under ContourCAM - Nest, and then the command Nest-1.



On screen you now have the rotated item and the original item. You are now to direct the rotated item to its final position.

1. Snap to the upper corner of the rotated item (point. 1).

2. Then snap to the upper corner of the original item (point. 2).

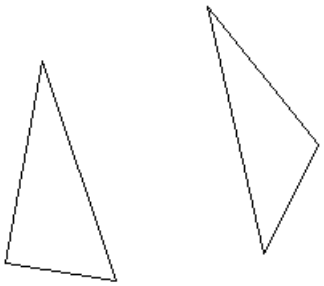
3. Set a point on the slanted line as a reference point (point. 3).

4. Set a point to the right of the slanted line to state the direction of the offset (point. 4).

5. Enter the offset = 2 mm.

6. Save the drawing and name it NT.1

7.1.8 Nest-2: Nest two different items to each other



1. Draw a triangle through the following points:

1. X30,Y30

2. X50,Y140

3. X90,Y20

4. X30,Y30

2. And one more through these points:

1. X140,Y170

2. X170,Y35

3. X200,Y95

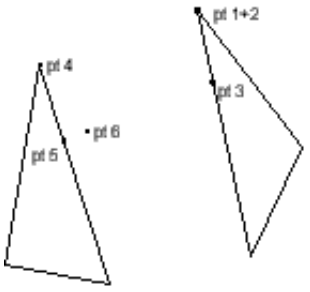
4. X140,Y170

3. Select the item to be nested by snapping to the upper corner of the last drawn triangle.

4. Now choose the command Nest-2

5. Now snap once more to the same point while this point later is to refer to a point you will set on the first triangle you drew.

6. Set a point on the reference line (point. 3), while this line later shall refer to a reference line on the triangle you drew first.

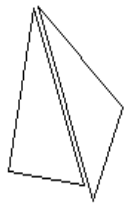


7. Snap a reference point to the upper point of the first drawn triangle (point. 4).

8. Set a reference point on the line (point. 5).

9. Set a point (point. 6) for the direction of the offset.

10. Enter 3 as distance between the two items.
11. Save the program and name it Nest.



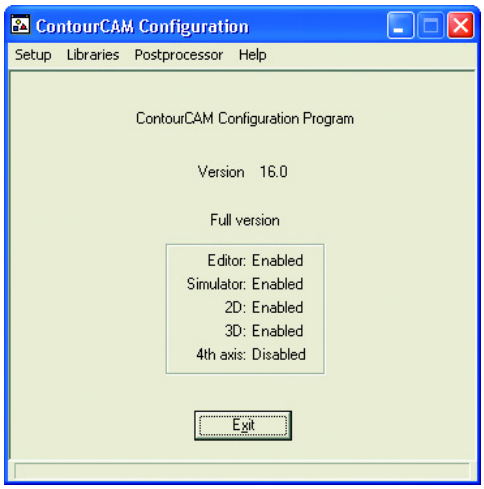
Above you see the two items after they have been nested to each other with a mutual distance of 3 mm. The points you set on each of the elements consequently refers to each other so in this way the nesting is carried out proportional to these points.

7.2 Defining tools and machine cycles

ContourCAM contains libraries, where you can define tools and machine cycles. To be able to generate the correct NC-code it is as an example necessary to know the exact measurement of the tool which is to be used for the actual preparation. In the following you are to define a tool and a drilling cycle. The procedure for defining tools as well as machine cycles is always the same why you should not have trouble setting up your own libraries once you examined the examples which follows:

7.2.1 Definition of tools

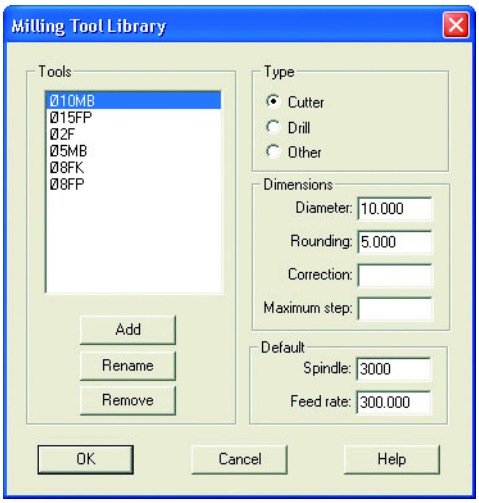
You have a tool - ball nosed cutter with a diameter of 10 mm - which you want to define for your tool library. From the menu ContourCAM in Windows you choose ContourCAM Configuration, whereupon the configuration dialog is displayed.

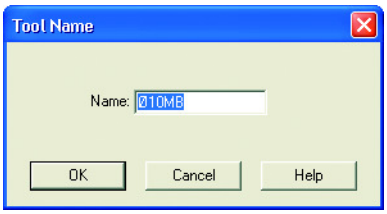


From the configuration dialog you choose Libraries and then Milling tool library whereupon the dialog in the figure is displayed.

Tool:

The window shows a list of all tools defined. Since you have not yet defined any tool your window will be empty. Now choose the flag Add since you want to add a tool to the library. In the dialog now appearing you are to enter the name under which you want to enter the actual tool to your list of tools. You are free to decide the name of your tool just be sure that the name you use is as unequivocal as possible so it is easy recognizable.





In this example we named the tool Ø10MB (diameter 10 - miller - ball nosed). If you want to another name just do so. When you have entered the name you click the OK button and return to the library dialog itself where you can see the tool you just defined has been added to your list of tools.

Type: With the mouse you click in the field just beside Cutter.

Dimensionns: Now you determine exactly what the tool looks like.

Diameter: Enter 10 (D10)

Rounding: Enter 5 (if you enter 0 as the rounding the tool is flat)

Displacement: Enter the possible value compared with the measuring tool.

Maximum step: Here it is possible to enter a maximum value for the infeed step valid for this particular tool.

Defaults: It is possible to define default values for the speed of the spindle and for the feedrate of the actual tool. If a tool is defined with default values those values will be transferred to the dialog, from where you call it automatically.

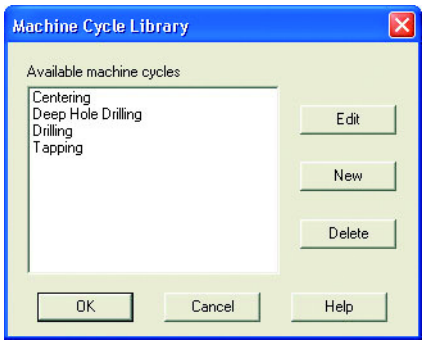
Spindle: Enter 3000 as speed of the spindle

Feed: Enter 300 as default feedrate

Click the OK button

Now you have defined a ball nosed milling tool with default values for speed of spindle and feedrate. Try to define some more of the tools you may have at your disposal.

7.2.2 Definition of a drilling cycle

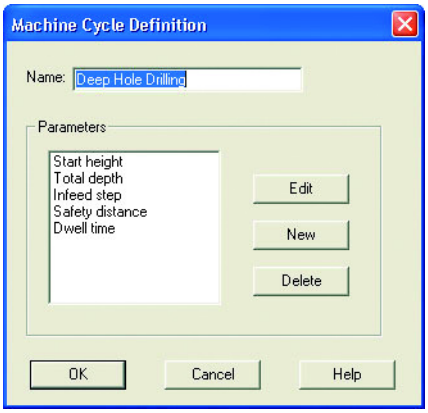


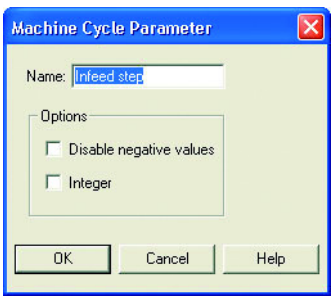
Now you are to define a drilling cycle. In the example which follows we have used a cycle Deep hole drilling from an ordinary Fanuc controller. From ContourCAM configuration you choose Libraries and then Machine cycles. Then the dialog in the figure appears.

Now you are to define a brand new cycle. Therefore with the left button on the mouse you click the field New. The dialog in the figure is displayed.

Name: Enter the name Deep hole drilling.

Now you are to enter the parameters relevant for the cycle you are now defining. Use the mouse to click the field New. Now a new dialog is displayed.



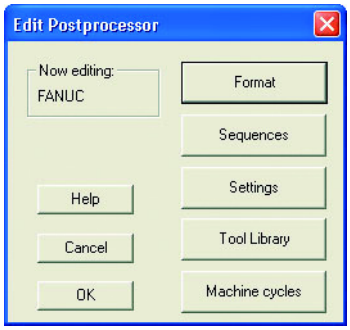


Name: Enter the name of the first parameter - in this case Start height. Click the field OK whereupon you return to the definition dialog itself.

Notice that the name of the parameter is automatically transferred to the list of parameters. Once more choose New and enter the name Total Depth as the next parameter. Now define the following parameters Infeed step, Safety distance and Dwell time following the same procedure exactly that you just been using.

When you defined all parameters click the OK field until you have returned to ContourCAM configuration.

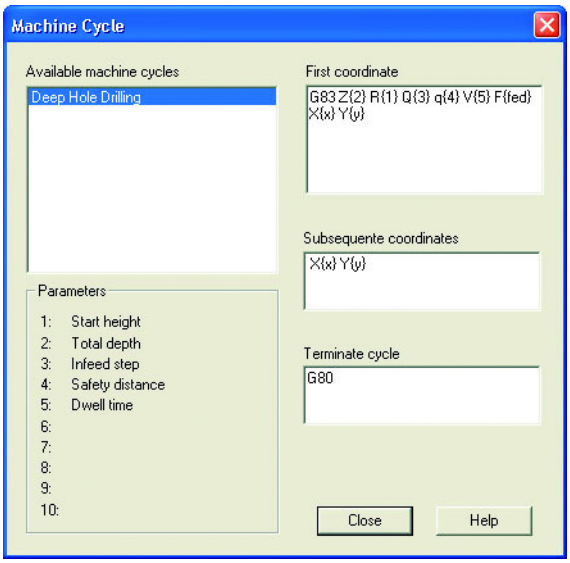
If you have another controller than the Fanuc used in the example above it is a good idea for you to define a cycle for your own controller. A defined cycle can be deleted by clicking on the name of the cycle to be removed (then it will be high-lighted) and then click the Delete field in the dialog. Try to define one or more cycles for your own controller.



Now you are to attach the defined parameters of the cycle to the Fanuc (or your own) post-processor where the definition of the cycle of the machine itself for carring out a Deep hole drilling. takes place. From ContourCAM configuration choose Postprocessor. Choose Edit and then click Fanuc (or your own controller) from the list of postprocessors. Now the dialog in the figure is displayed.

Click the field Machine cycles, and a new dialog appears.

As you will notice the cycle you have just named has been transferred to the list of machine cycles of your postprocessor. In the windows to the right First coordinate, Subsequent coordinates and Terminate cycle you now enter the parameters of the controller needed to carry out the actual cycle. Try to define more drilling cycles yourself.

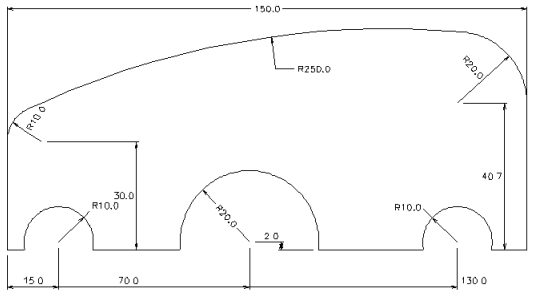


7.3 Examples for NC generation

7.3.1 Preparation of a drawing for NC generation

In this example you will learn how by using ContourCAM you define a geometry from a concrete task. We will follow the performance from receiving a drawing from a customer, until you are ready to transmit the final NC-code to the controller.

7.3.1.1 The construction phase



In the figure you see the task you are to solve. You decide to start in the coordinate 10,0. State the origin in the coordinate 10,10.

1. Draw a line (<Ctrl>+V) through the points 0,0 and 150,0.

2. Draw a circle with radius 10 and center in 15,2.

3. Draw a circle with radius 20 and center in 70,2.

4. Draw a circle with radius 10 and center in 130,2.

5. Draw a line from 150,0 to 130,45.

6. Draw a circle with radius 20 and center in 130,40.75.

7. Draw a circle with radius 10 and center in 10,30.

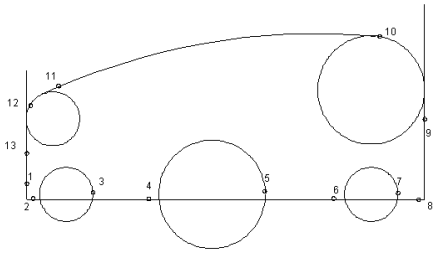
8. Draw a vertical line (<Ctrl>+V) starting at 0,0 and make sure it is sufficiently long.

9. Choose the Fillet-2 command from the Edit|Trim menu. Set the first point at the top of the circle to the upper right. Set the second point at the top of the circle to the upper left. For center of the round off set a point in the middle of the drawing, and enter a radius of 250.

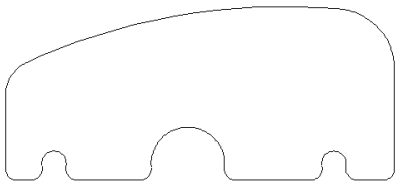
As you have seen during the construction we did not care about the intersection points between the different entities only you drew complete circles and lines a bit to long. You are now to remove those parts of the elements which are not part of the final item. For this purpose you use the command MagicTrim.

Points to set when MagicTrimming.

1. Choose the command MagicTrim from the menu Edit - Trim/Fillet. You are now asked to set points on those parts of the entities to be represented in the final item. Note It is very important that the points are placed as close as possible to the intersection point with the following entity.



2. Set the points from 1 to 13 as shown in this illustration.



After MagicTrim.

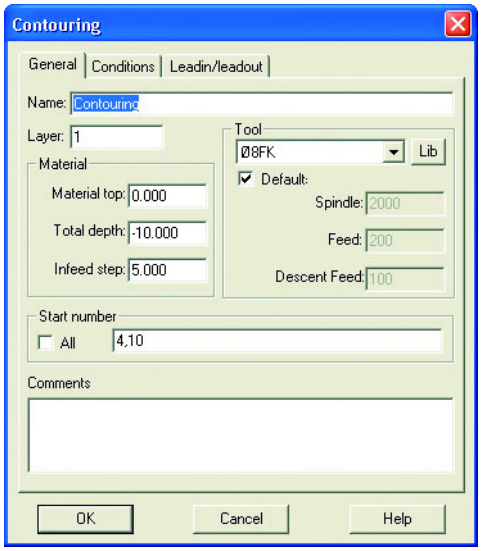
3. When the points are sat. Click the central mouse button or press <Enter>

MagicTrim now removes all superfluous entities and leaves the final item on the screen. EASY DON'T YOU THINK!

4. Save the drawing (<F12>) and name it T1.

7.3.2 Generating the NC-code

You are to mill the contour of your drawing and now you are ready to generate the NC-code. Under ContourCAM from the main menu you choose the command Milling and then Add. From the menu that now appears you choose Contouring. A new dialog is displayed.



Name: Accept the name Contouring.

Layer: Enter 1 for layer No. one.

Material top: Enter 0 as the top of your material.

Total depth: Enter -5 as the total depth.

Infeed step: Enter 5 as the infeed step per each path.

Tool: From your list of tools you choose a tool with a diameter of 10 for milling ball nosed or some other tool you already defined.

Spindle: Enter speed of the spindle 3000

Feed: Enter feedrate 300.

Descent feed: Enter 100 as the feedrate for vertical movements into the material.

Start number: Since this is contouring and since you are not to do further preparation for this item you do not have to state start number 1 because there are no other start numbers in your drawing.

Comments: If you wish to make comments you are free to do it inside this window.

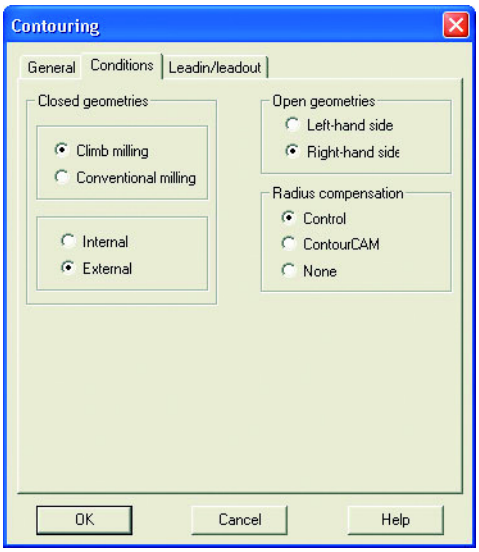
When you entered the general information for the item to be prepared you click the flag Conditions where you are to determine, how you wish the preparation itself to be executed. Now the dialog in the figure are displayed:

1.Click the field to the left of Climb milling. When the field is active it is market with a black dot.

2.Click the field External as the contouring must take place on the outside of the item.

Since it is a closed contour the questions in the dialog about open contours does not have to be answered, since they have no influence on the actual item.

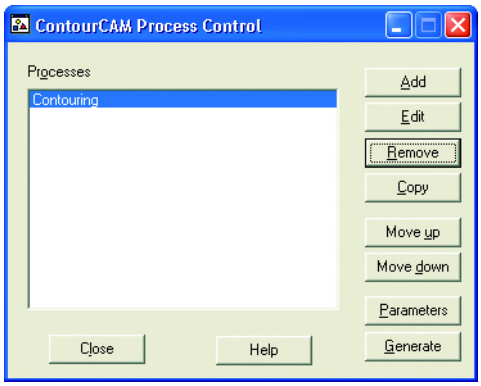
3.Compensation of the tool. Click the field Control since you wish the controller to make the compensation of the tool.



4.In this case you want a linear lead in to the item, so you click the field besides Linear. The length of the lead in must be 10 mm. In the window Length enter 10. Angle: enter 0 since the lead in line differs from the first line in the drawing to be milled 0 degrees.

5.Repeat the procedure making the lead out and use the same length 10 mm.

Accept what you keyed in by clicking OK. Now you return to ContourCAM Process Control.

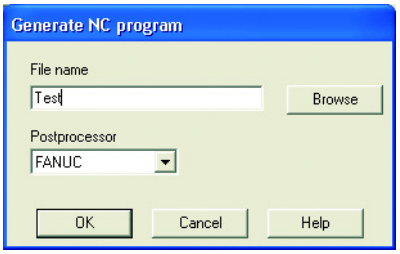


You are now to let ContourCAM generate the NC-code according to the information stated in the dialogs. Click Generate. Now the dialog in the next figure appears.

Filename: Enter Test as filename.

Postprocessor: Choose the postprocessor wanted from the list.

Click OK, after which ContourCAM generates the program. In a little while ContourCAM switches to the editor, where the NC-code of the generated program is displayed..



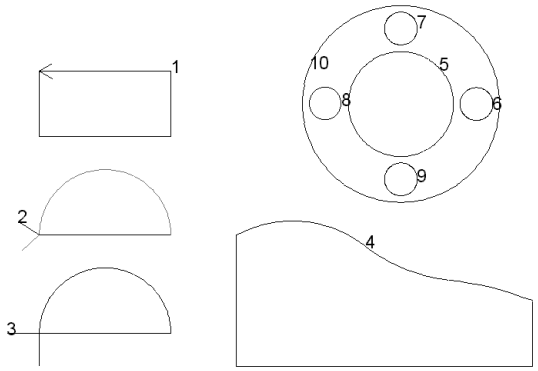
7.3.3 In ContourCAM's editor

You now have the possibility to add change or edit the program (see chapter 2) before it is send to the CNC-unit.

When the program is ready you choose Cam from the menu and then Send program. The program will be transmitted to the CNC-unit connected.

7.3.4 Cutting priorities generating NC-code

Adjust your screen to be 250 units wide (Ctrl+4/Ctrl+6)



Drawing exercise in cutting priorities.

In this lesson we shall make a drawing as shown in this illustration. The purpose of this exercise is to show how to decide which direction ContourCAM should go. For the left part of the drawing we will use the toolpath process while using ContourCAMing for the right part.

1. Draw toolpath 1

Draw a rectangle using the line command (V) with lower left corner at coordinate 20,40 and upper right corner in 40,50. It is important that each side of the rectangle is a separate line. Draw the line between 20,50 and 40,50 as an arrow (>) as shown in the illustration above. Write the text "1" and snap the corner of the rectangle as shown in the drawing.

ContourCAM will read the drawing as follows: Go to start position no. 1. Since two entities branch off from the start point ContourCAM will select the entity with the highest priority, in this case the arrow as it has higher priority than a line.

2. Draw toolpath 2

Draw a line from 20,40 to 40,20. Now draw a red semicircle as shown in the drawing (press H to change color). Draw two line segments starting at coordinate 20,25 as shown (draw the lower line using red and the upper black). Snap the text "2" to the upper line.

ContourCAM reads this part of the drawing as follows: Go to start position 2. Follow the first line to the intersection with the other entities (Observe that only intersection of end points count). From this point three entities branch off, as both the semicircle and the lead out have a lower priority color than the vertical line (black have the highest priority) the route is determined.

Draw a line through the following points: 20,40 and 40,20. Then draw a semi circle (use color red (H)), as shown in figure no. 2 in the drawing. Draw to line segments (the lower of the two lines must have a lower color than th upper line e.g. blue) starting in the coordinate 20,25 fairly as in the drawing. Attach the text 2 to he endpoint of the upper line,

ContourCAM reads the above as follows: Go to start position no. 2. Follow the first line to the intersection with the other entities. From the intersection point three different entities branches off. The semicircle and the lead out line both have a color lower than the horizontal line the toolpath is determined.

3. Draw toolpath 3

Draw a line between the coordinates 20,10 and 40,10. Draw a semicircle as shown in the figure. Draw the two lines representing lead in and lead with a lenght of 5mm each. Attach the text 3 to the endpoint of the horizontal lead in.

ContourCAM reads the above as follows: Go to start position no. 3. Follow the first line to the intersection with the other entities. Contrary to the previous example all entities have the same color. Since circles has a higher priority than lines the toolpath automatically takes the circle before the lines. The route is determined.

4. Draw a Contour

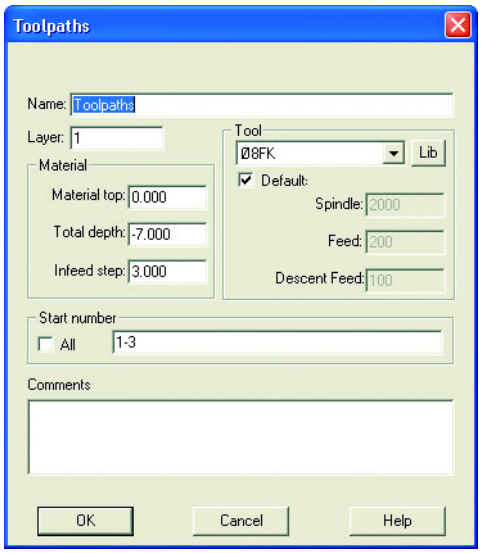
Draw a line between the coordinates 50,25 - 50,5 - 95.5 and 95,15. Draw a circle R25 and center in the coordinate 85,43. Using the command Fillet to point (Edit/Trim-Fillet) you now draw the upper fillet R18 and the lower with fillet R50. Trim the entities until you have a figure like the one in the drawing. Attach the text 4 to the intersection point of the two upper arcs.

When milling a Contour the lead in and out are determined in the dialogs for contouring.

5. Draw one more contour

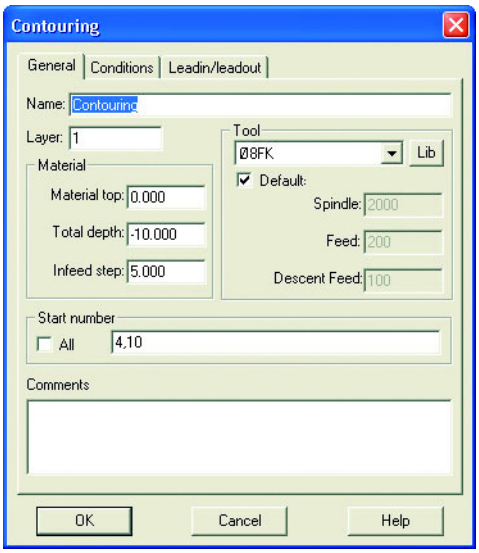
Draw a circle R8 and center in the coordinate 75,45. Draw one more circle R15 with the same center. One more circle R2.5 with center in the coordinate 86.5,45. Select the circle R2,5 and use the command Circular array under Edit/Selection edit to draw three copies of the circle. Attach the numbers from 5-10 as shown in the drawing.

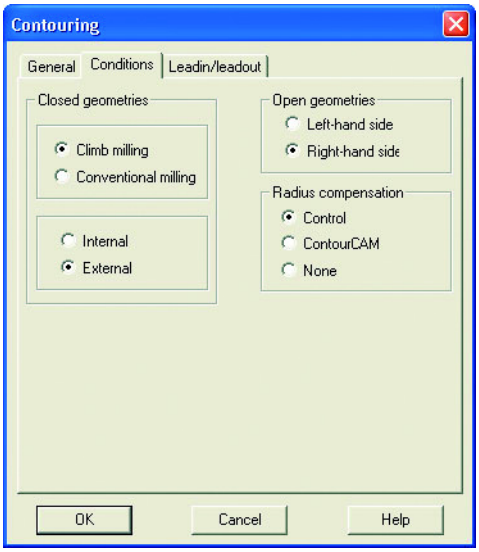
When your drawing looks like the drawing beginning this task you are ready to generate NC-code of the whole drawing. Choose milling and then the command toolpath. Fill in the dialog as shown below.



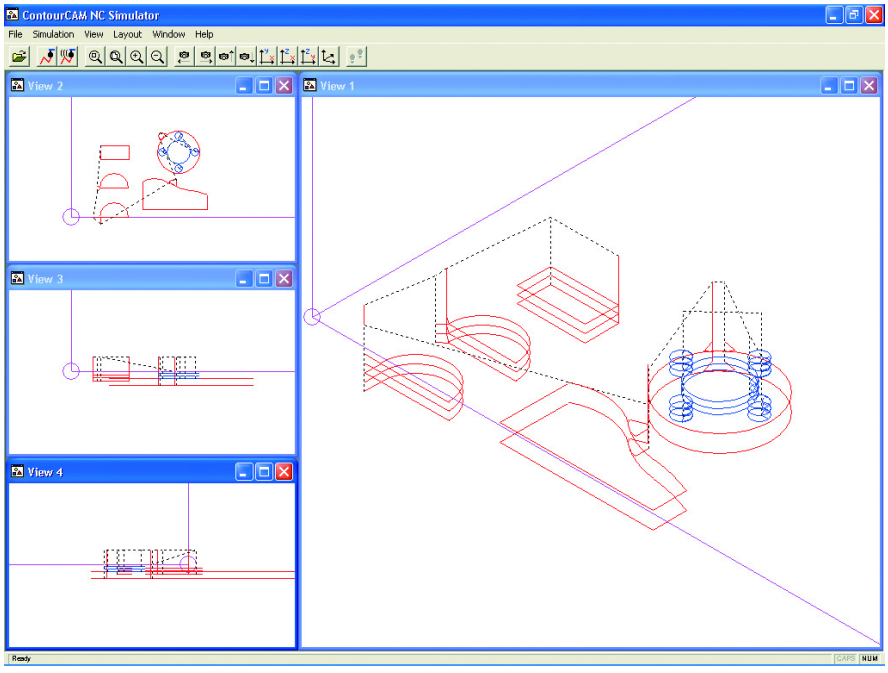
When you have completed the dialog for the toolpath you now choose contouring under Add from the process control. Then you fill in the dialogs as those below:

As it appears in the dialog for the general adjustments it only applies the start numbers 4 and 10. You now choose one more contouring and repeat the process for the remaining start numbers filling in the dialogs as shown here:





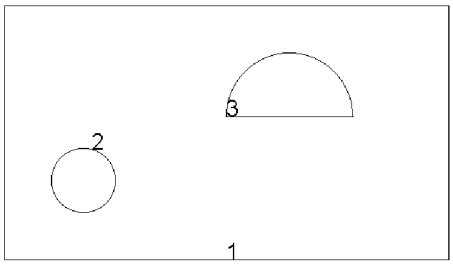
When all dialogs are completed you choose Generate from ContourCAM Process Control entering T1 as the name of the program and select your postprocessor. In ContourCAM's editor you now check, if the working process was performed as intended.



7.3.5 Pocket milling: Advanced

Now you are to make a pocket milling where a couple of islands are included.

1. Draw a line through the points X30,Y30 and 30,110
2. Draw one more line through 30,110 and 170,110
3. And one more line through 170,110 and 170,30
4. Draw a line through 170,30 and 100,30
5. Snap from the endpoint 100,30 to the starting point 30,30
6. Draw a circle with its center in 55,55 and a radius of 10 mm.
7. Draw a line through the points 100,75 and 140,75
8. Draw a semi circle from 100,75 to 140,75
9. Press T (text) and write the text 1, snap the text to the point 100,30
10. Write the text 2 and snap it to any point on the border of the circle
11. Write the text 3 and snap it to the point 100,75

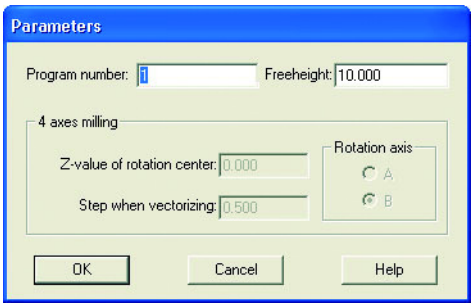


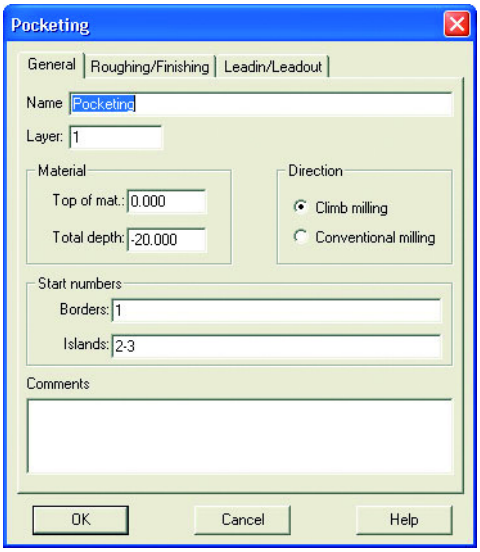
Your drawing should look like this

Choose ContourCAM from the main menu and then Milling. The process control in ContourCAM is now displayed. From here you choose Parameters after which the parameter dialog appears.

1. Enter 1 as the program number
2. Enter 10 for the height where movements in rapid traverse are allowed.
3. Click the OK button

Now you are back in the process control dialog where you click the Add button. Now choose pocket milling. A dialog like in the figure below is displayed.





Name: Accept the name pocketing.

Layer: Enter 1 since the drawing is placed in layer no. one.

Material top: Enter 0 as the value in Z for the top of the item.

Total depth: Enter -20. In this case the minimum Z-value of the item.

Direction: Click the field Climb milling. Notice that the field next to climb milling now is marked with a black dot to indicate that it is now active.

Borders: The border of the area to be cleared has number 1. Enter 1 for the border.

Islands: The islands to left is numbered 2 and 3. Enter 2,3.

Now you have filled in the dialog with general information for the course of the preparation. Now click the flag Roughing since now you are to determine how you want to do the roughing. When this is done the dialog in the figure is displayed.

Tool: From your list of tools you choose a tool with a diameter of 6 or some other tool you already defined.

Spindle: Enter speed of the spindle 3000

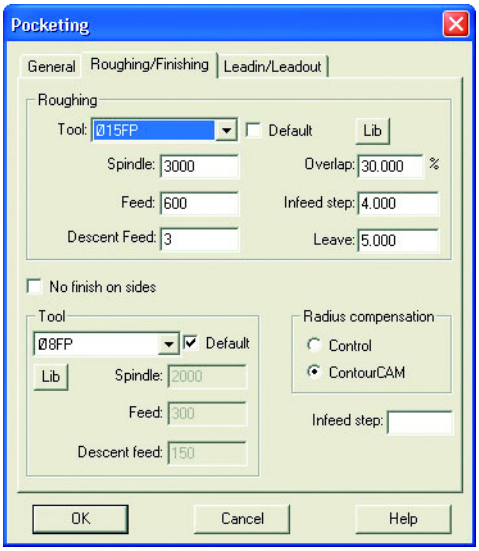
Feed: Enter feedrate 200.

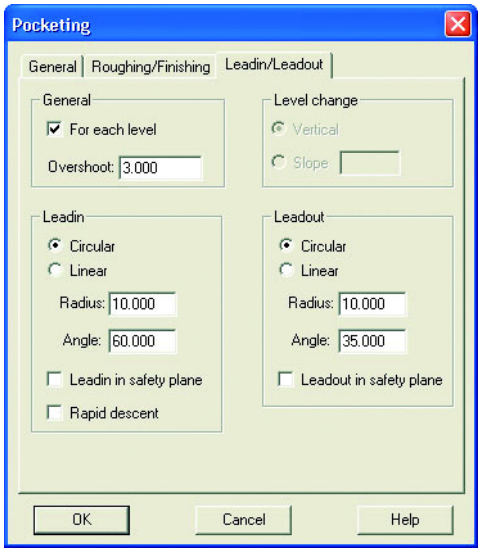
Overlap: Enter 30, which is the overlap you wish for each path.

Infeed step: Enter 5 for the maximum step in Z.

Leave: Enter 1. By this you state that you want 1 mm of material left for the final cut.

Now fill in Finishing to determine how the finishing should be made. A new dialog is now displayed.





Tool: Choose a tool with a diameter of 5 (or a similar tool) from your list of tools.

Spindle: Enter speed of spindle 2000

Feed: Enter 200 for the feedrate

Radius compensation: Click the field besides Control, since in this case you want to use the compensation of tool on the controller.

Infeed step: Enter 5 as maximum infeed step per path.

Overshoot: Enter 3. You wish a distance of 3 mm between the lead in and the lead out to avoid that any material is left.

Lead in: Choose circular lead in by clicking the field besides Circular. Enter 3 for the radius of the lead in and 45 for the angle of the lead in arc.

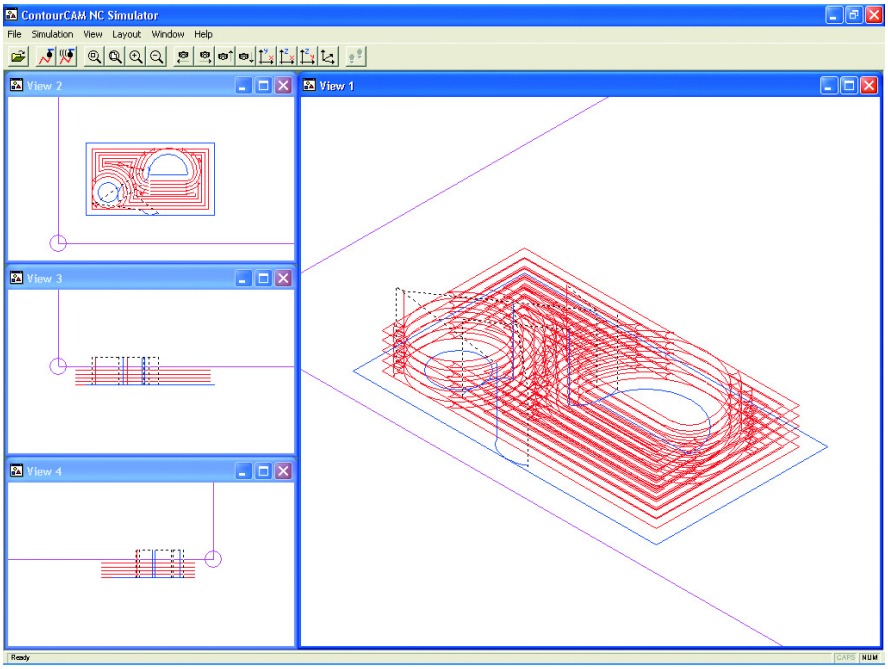
Lead out: Enter 3 for the radius an 35 for the angle of the lead out.

Now click the OK button, after which you will return to the ContourCAM Process Control. From here choose Generate. In the next dialog you enter the filename Poc and choose a postprocessor from the list.

7.3.8 In the Editor of ContourCAM

It is now possible to add - change or edit in the program (see chapter 2) before sending it to the CNC-unit.

When the program is ready it is possible to choose Simulate NC-program under Cam in the main menu, which graphically shows, how the preparation will be performed.

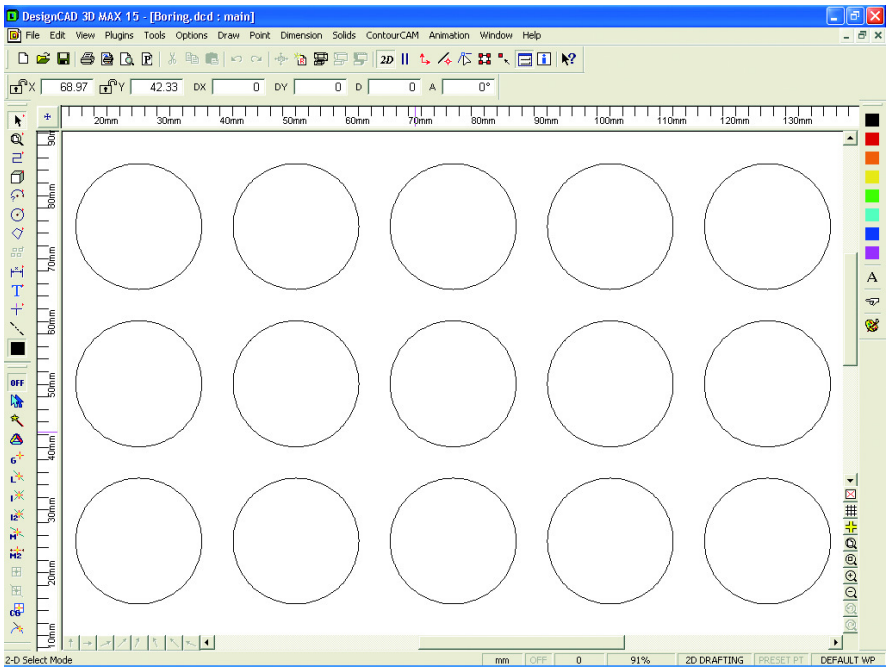


If you choose the command Send program, the program will be transmitted to your CNC-unit. Try to make some other examples of pocket milling on your own.

7.3.9 Machine cycles

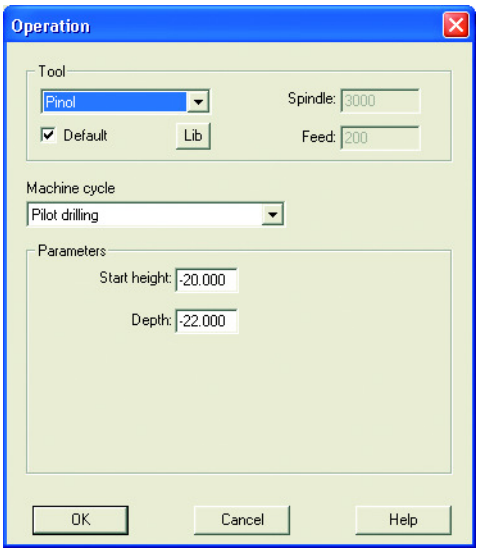
In this chapter we shall see how it is possible to make use of the cycles available in the controller of your machine and possibly to call those cycles during the phase of construction.

1. Draw a circle radius 10 with the center in the coordinate X25,Y25
2. In the tool box you choose array ordering and select the circle you just drew. Press enter.
3. In the field in the upper left corner you enter 5 as number of copies.
4. Then enter 3 as number of rows.
5. Click the field Use original as first copy (active when marked with a cross).
6. Set a point in the coordinate X50,Y25 for the placement of the second copy.
7. Then set a new point in the coordinate X25,Y50 for placing the rows.

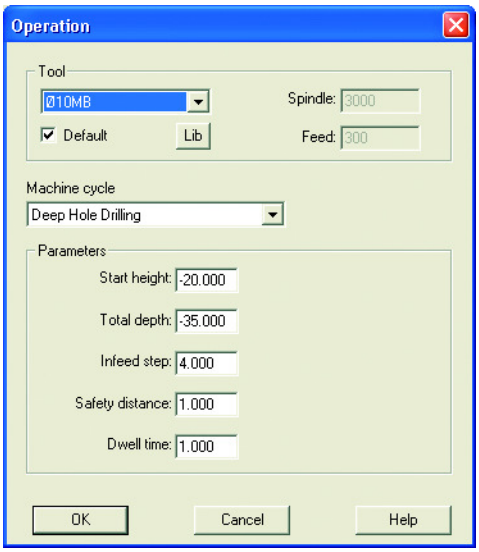

DesignCAD 3D MAX 15 - [Boring.dcd : main]
File Edit View Plugins Tools Options Draw Point Dimension Solids ContourCAM Animation Window Help
X 68.97 Y 42.33 DX 0 DY 0 D 0 A 0°
2-D Select Mode
mm OFF 0 91% 2D DRAFTING PRESET PT DEFAULT WP

Now the screen shows a drawing as the one in the figure. Now imagine that the holes must be prepared as follows. First operation should be a pilot drilling then you want to make a deep hole drilling and at last you are to prepare for a tapping (be sure you have defined the cycles necessary before making this exercise).

Under ContourCAM from the main menu you choose Milling and then Add. Click Boring etc. and the dialog in the figure is displayed.



As the dialog shows ContourCAM has already observed the size of the holes (D20). Now you are to define the operations to be executed on the given holes. Click Add and a new dialog will be displayed.



The first operation to be executed must be a pilot drilling. First choose the tool you wish to use for the operation in question (in this example the tool is named Pinol). If you have defined your tool for pilot drilling and saved it under another name you just select this tool.

1. From your list of tools select the tool meant for pilot drilling.

2. Enter speed of spindle and feedrate.

3. From the list of machine cycles choose pilot drilling.

4. Enter start height 0.

5. Enter the depth of the drilling 2 mm

6. Click OK

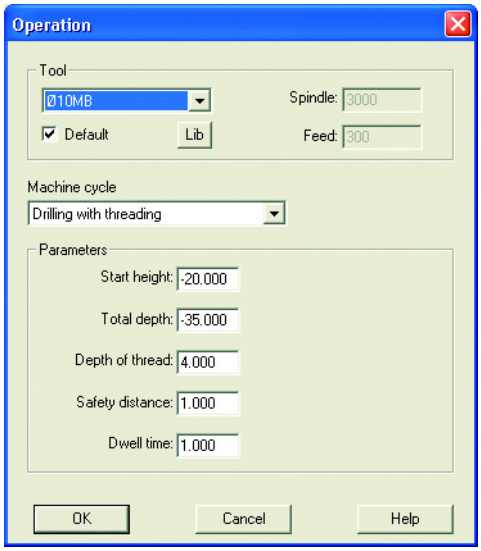
Now you have defined how you want the pilot drilling to be made on all holes. From the dialog choose Add again.

1. Choose the tool to be used for the deep hole drilling (D20)

2. Enter speed of the spindle and feedrate

3. From the list of machine cycles now choose Deep hole drilling (or the name you applied for the corresponding operation).

Now the list of parameters needed for this operation will be displayed (corresponding to the parameters you stated when you defined this particular machine cycle). In this example the relevant parameters appear as the dialog below:



1. Enter start height 0

2. Enter the total depth of the drilling -20

3. Enter maximum infeed step per drilling 4

4. Enter safety distance (the height where movements in rapid traverse are allowed) 1 mm

5. Enter dwell time 1

6. Click OK

Now you defined a deep hole drilling. The last operation to be carried out is the tapping.

1. Choose Add from the dialog Boring.

2. From the list of tools choose the tool to do the tapping.

As you can see there are three different parameters to carry out the operation (on the machine used in our example). In this case you want a tapping from height -10 to the bottom of the hole:

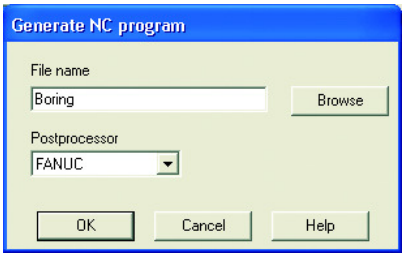
3. Enter start height -10

4. Enter total depth for the tapping -20

5. Enter dwell time 1

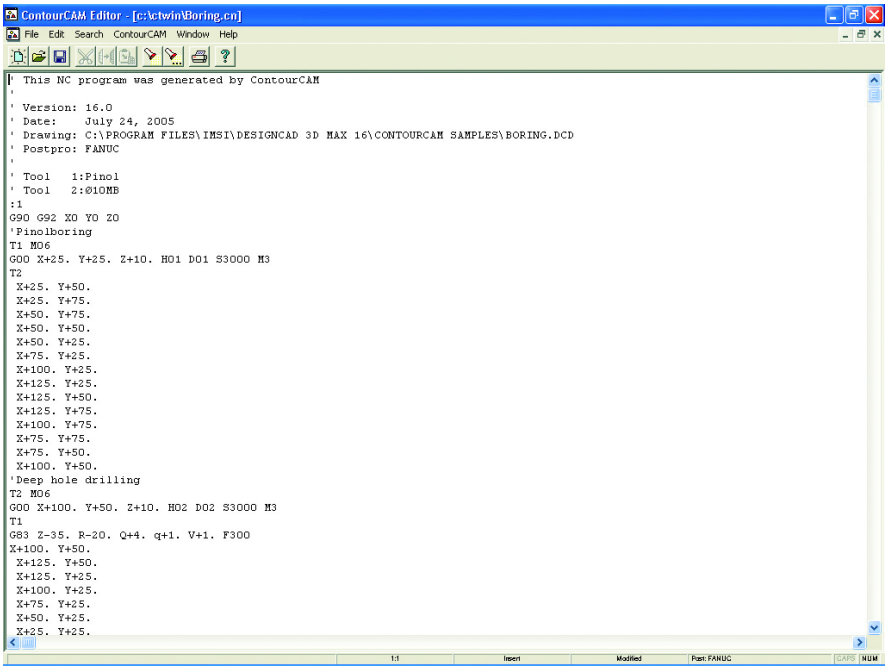
6. Click OK

Now you have stated the parameters for the tapping. In this exercise you were to prepare three operations for each of the holes in the drawing. This is now done. From the dialog Boring you click the field Generate. Now the dialog below appears:



1. Enter the name the NC-program
2. Choose a postprocessor and click OK

Now ContourCAM calculates and generates the hole program in consideration of the machine cycles chosen and deliver the final program in the editor ready for possible editing or direct transmission to the CNC-unit.



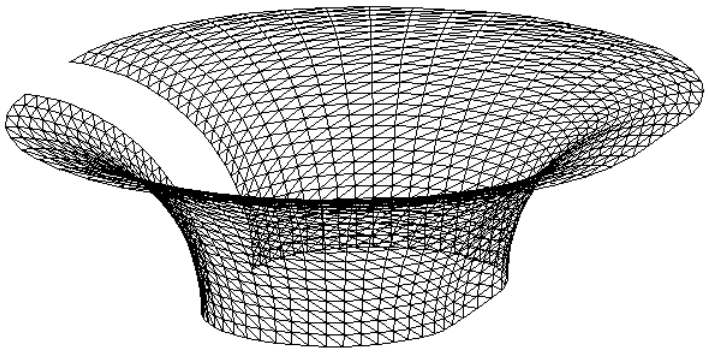
In the next chapter we present some examples to make you more confident in 3D programming. There are examples in snake programming in DesignCAD 2D starting with very simple tasks and including engraving of texts and logos.

Tutorial ContourCAM 3D

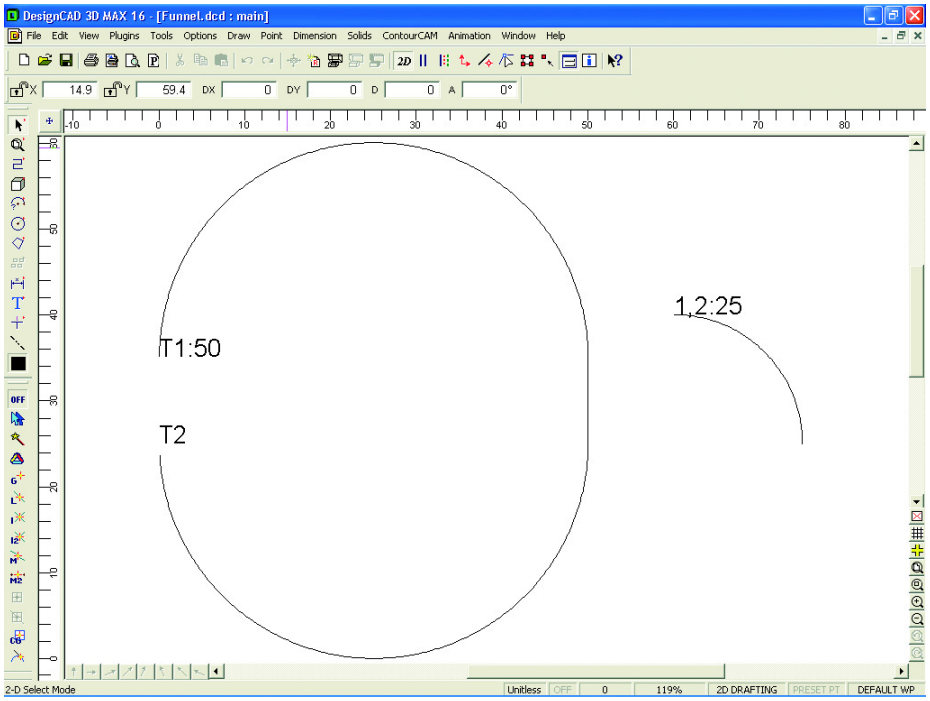
8

8.1 Your first snake

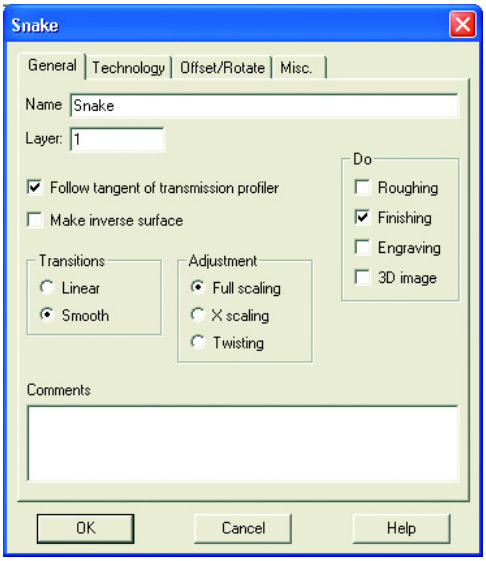
In this first lesson we will make an NC-program for milling a surface as shown below.



We will make an NC-program for milling this surface in lesson 1. At first glance this looks like a complex 3D surface that we need a 3D CAD system to represent. However, as we shall see, it is actually quite simple to make using a 2D drawing and ContourCAM's snake system.

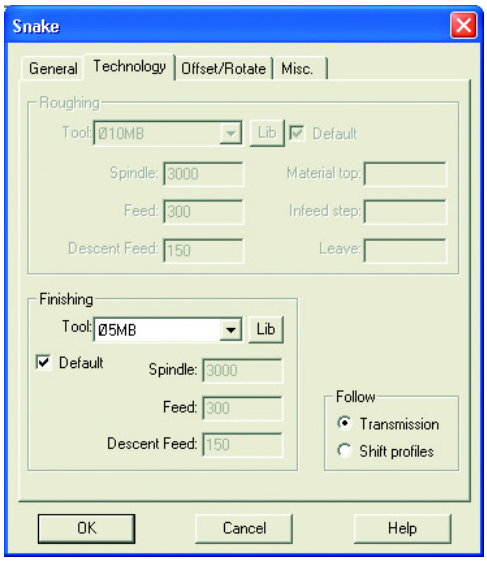


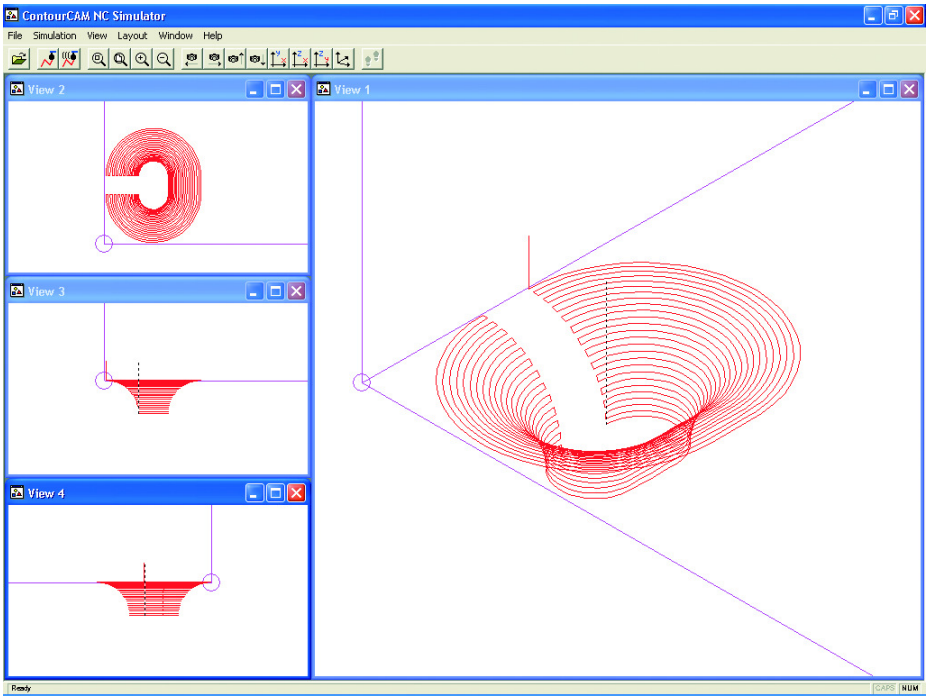
Draw a semi circle from point (0,10) to point (50,10), and one from point (50,0) to (0,0). Draw a line from point (50,0) to (50,10). This geometry is what we call the transmission profile. Draw a quarter circle from point (60,15) to point (75,0). This will be the shift profile. Now we need to add the texts. Write the text “T1:50” at point (0,10). The T indicates that this is a part of the transmission profile. The one that we want the shift profile number one to be attached here. Finally, the :50 means that we want the geometry to be divided into 50 line segments from this point to the next text. At the coordinates (0,0) write the text “T2”. This indicates that this is the point of the transmission profile where we want shift profile number two attached. Since this is the end of the transmission profile there is no indication for division as in the first text.



Write the text “1,2:25” at point (60,15). This means that this is both shift profile one and two. The “:25” means that the profiles should be divided into 25 line segments. Now we are almost ready to define the NC-program, but first we should save the drawing. From the menu Files select Save As, and name it “lesson1”. Select Milling from the menu ContourCAM. Press Add in the dialog that appears and select Snake. A dialog as shown in ? now appears. Set the input as shown. In this lesson we will only generate code for the finish path.

Select a ball headed tool with a diameter around 5. When you press the tab labeled Technology the contents of the dialog changes to what is shown in ?. All input fields related to roughing are disabled as we chose not to generate a roughing program. For the finishing we select a ball headed tool with a diameter of approx. 5 mm. Press the button in the finishing group and select a tool fitting the above description. If there is no such tool available in the tool list press the button and create a new tool. We want the toolpaths to primarily follow the transmission profile since this will result in mostly horizontal movements. The contents of the other pages of the dialog is of no interest to this assignment. Press the OK button the accept the input.

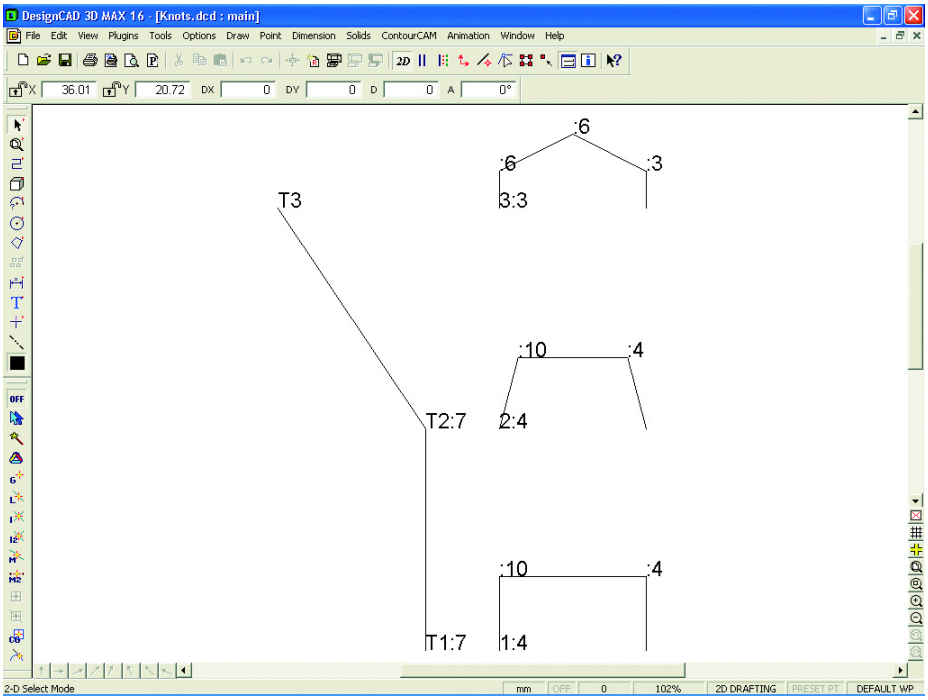




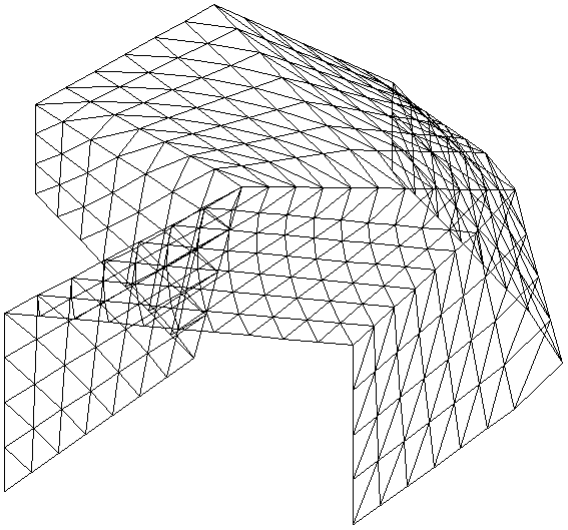
All we need to do now to get the NC program is to press the Generate button and enter a filename. ContourCAM automatically proposes the name of the drawing to be the file name for the NC program. When you press OK the program will be generated and loaded into the editor. Pressing F7 will start the simulator and show a program as in ?.

8.2 Using knots

In this lesson the matter of individual division will be revised. This will often be necessary when there in the surface occur sharp corners, like in this assignment.



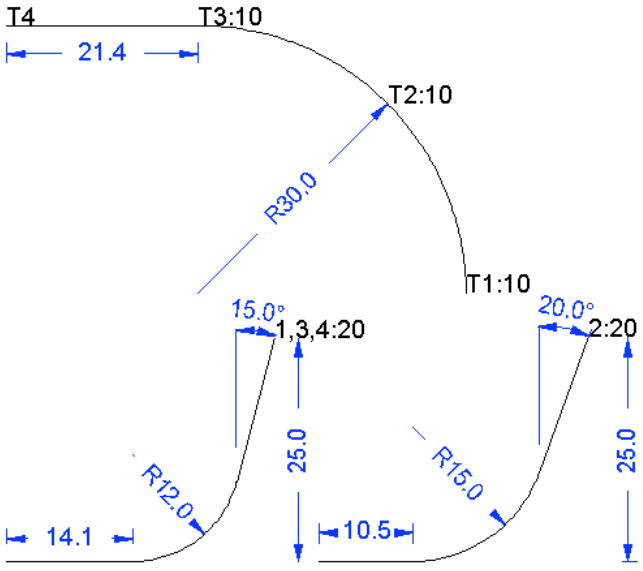
If you do not divide the shift profiles individually changes are the corners will be cut which is probably not desirable. The snakes require there to be the same number of line pieces in each shift profile. If this is not the case the snake command will display an error message informing you in with shift profile the problem is. How to make the division into line pieces must of course be decided from the drawing or the surface. But the main rule is: If you want a tool path for every 2 mm then the number of line pieces is the length of the profile divided by 2.



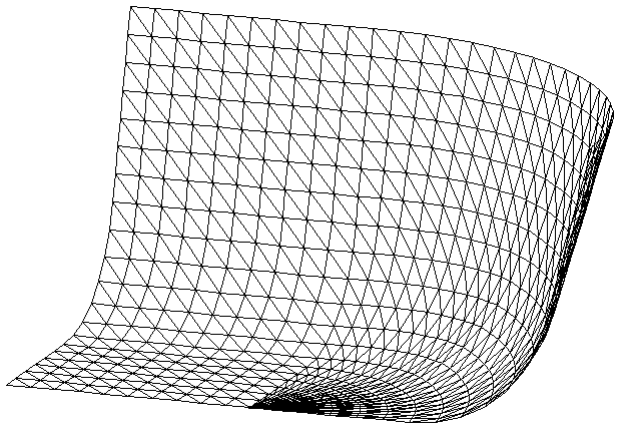
In this case there are, in the first two profiles 4+10+4 = 18 line segments. In the last shift profile has a line (entity) more so here the division is 3+6+6+3 = 18 line segments. Observe that the distances between the knots are approximately constant which is the objective. Make a drawing similar to that shown in ?. Do not be concerned about the exact dimensions; the objective is to get to know the principals. Activate the milling command and add a snake process. Select linear transformation, this way the effect of placement of knots. If you check the 3D image option and enter a file name on the forth page (Engrave), ContourCAM will generate a 3D drawing (.dcd) as shown here..

8.3 ISO corner

In this lesson there are three shift profiles in the snake. The surface is extraordinary since the radius must be constantly R=50 in horizontal intersections. It is therefore necessary to program a shift profile in 45?, that is less steep than the two profiles in 0? and 90?, respectively. Because the radius must be 50 mm all the way down to the arc vil afstande i 45? øges i forhold til 0? og 90? i en given højde. >>> Dette fatter jeg ikke en lyd af <<<The shift profile in 45? has been calculated and is shift profile no. 2.

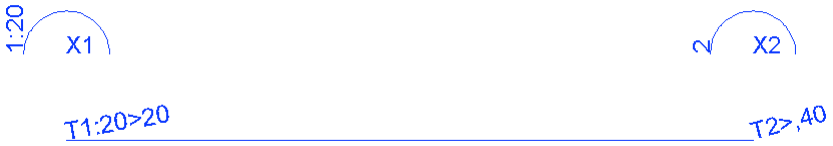


Observe, you need to divide the transmission profile into two 45? arcs, since shift profile can only be attached to end points of entities. To manage three shift profiles simultaneously and ensure a smooth transition from shift to shift can be a problem. If you touched the surface at the middle shift profile you would feel an edge if ContourCAM did not take this into account in the programming. By setting the general parameters you decide if you want linear or smooth transitions between the shift profiles; for this surface you should select smooth transition. If you select linear you will get an edge at the second shift profile which you can feel with a finger. This happens because the two knot on each side of shift two in the tool path being generated does not have tangent through the shift knot.



To generate this figure and see it in the simulator follow the same procedure as in lesson 1.

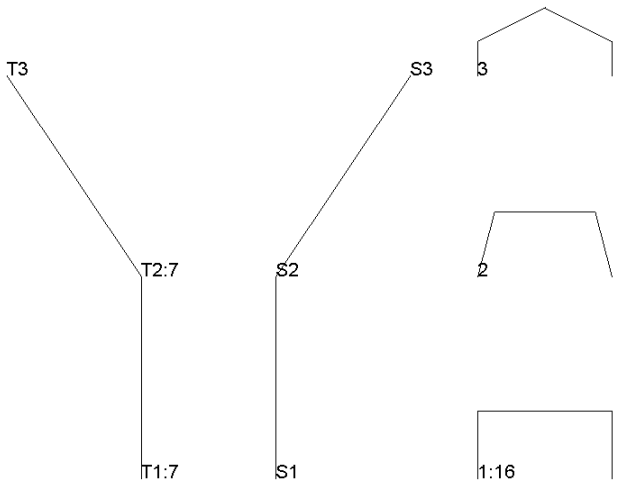
8.4 Rotating Shift Profiles



A snake with rotated shift profiles.

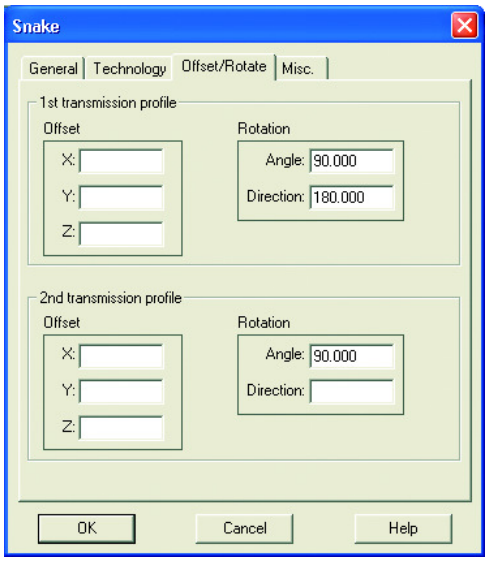
In this lesson we give examples on the slightly more complicated codes for rotating the shift profiles relative to the transmission profile . Using the code T1:20>20 the shift profile labeled 1 will be rotated 20° relative to the transmission profile. Using the code T2>,40 the shift profile labeled 2 will be rotated 40° relative to the normal vector of the transmission profile.

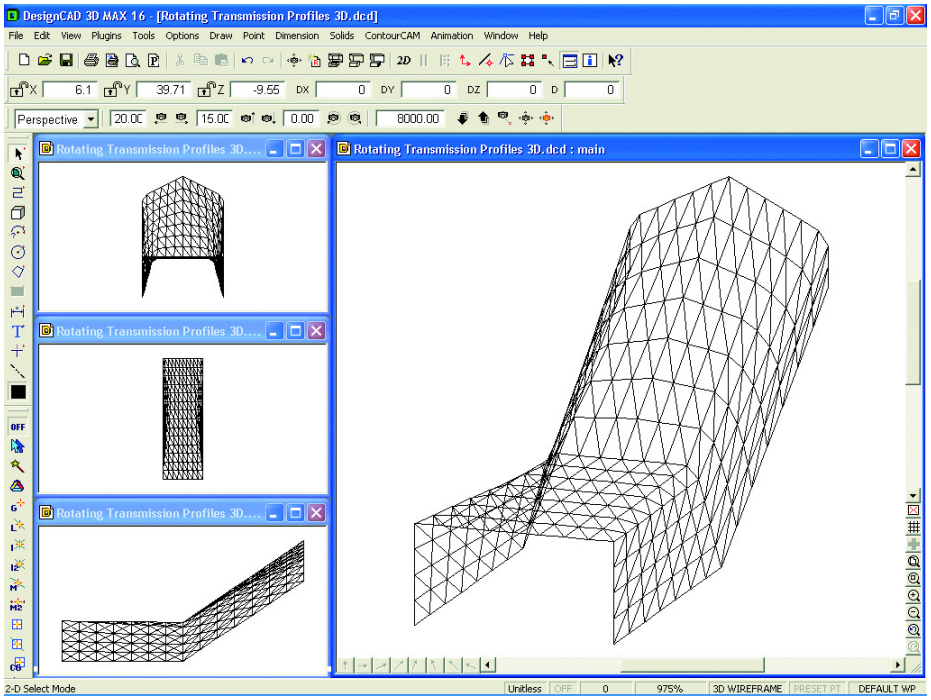
Naturally, these codes can be used simultaneously; for instance, you could have the following code: T1:20>20,40. It is plain to see how the shift profiles have been rotated relative to the transmission profile. Again do not be concerned by getting the correct dimensions but make sure you understand the principal.



8.5 Rotating the Transmission Profiles

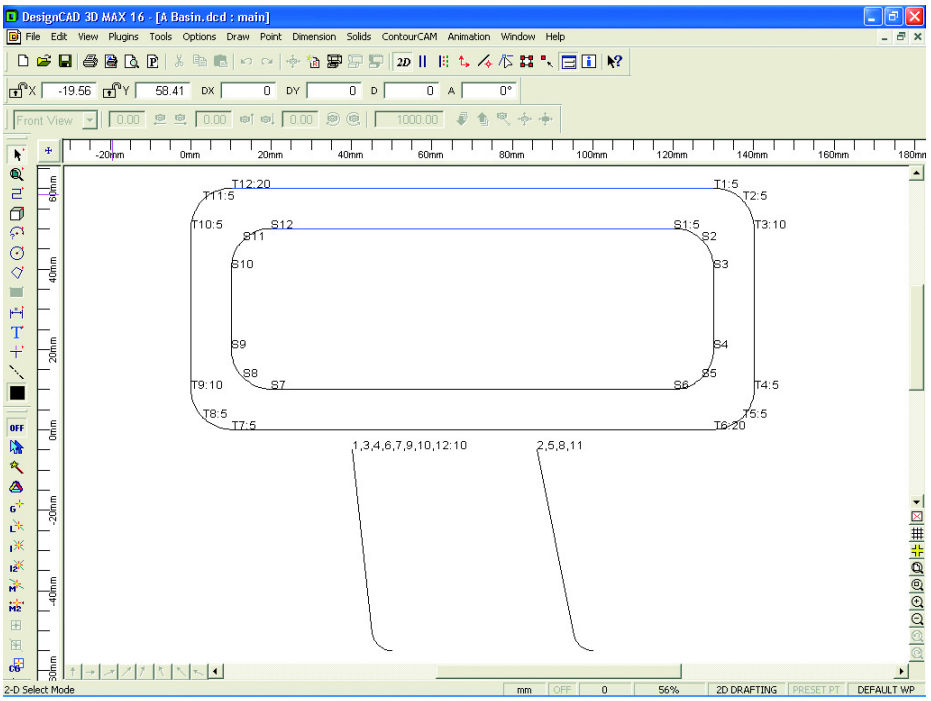
Here we want to rotate the transmission profiles. In this lesson we will see how to make transmission profiles that do not lie in the XY plane. When they have been rotated the two transmission profiles will be parallel, which is the cause for the shift profiles to have the same width. Make a drawing similar to that shown in ??. Activate the milling command and add a snake process. On the first uncheck the option Follow tangent of transmission profile. This makes the shift profiles stand vertical. If this option is checked they will be rotated according to the tangent of the transmission profiles.



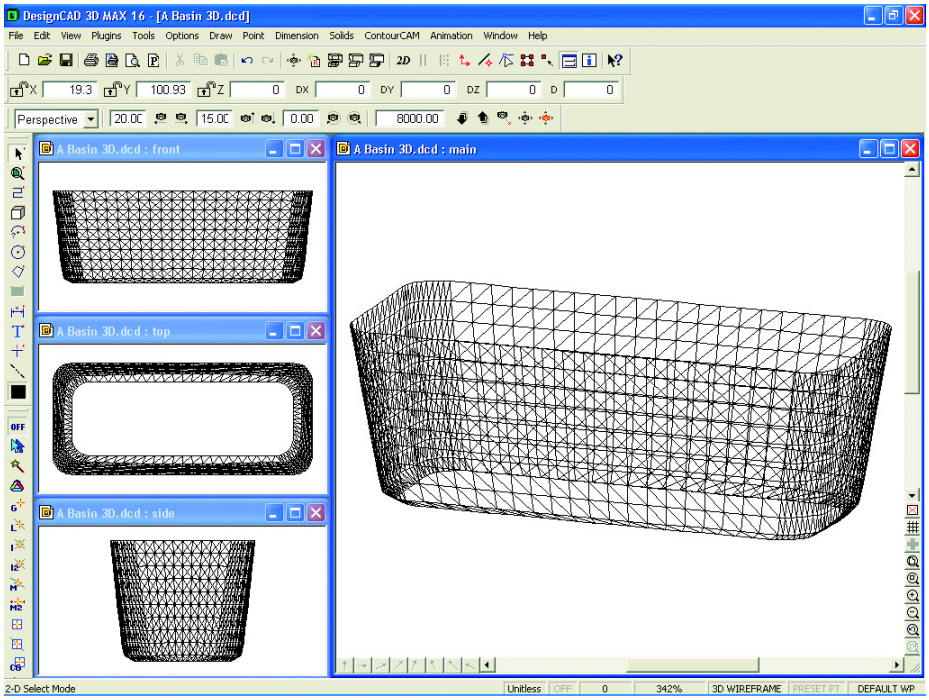

DesignCAD 3D MAX 16 - [Rotating Transmission Profiles 3D.dcd]
File Edit View Plugins Tools Options Draw Point Dimension Solids ContourCAM Animation Window Help
Perspective
Rotating Transmission Profiles 3D....
Rotating Transmission Profiles 3D....
Rotating Transmission Profiles 3D....
Rotating Transmission Profiles 3D.dcd : main
2-D Select Mode
Unitless
3D WIREFRAME
DEFAULT WP

We rotate the first transmission profile -90° and the second 90°. The rotation of the transmission profiles is defined on the page labeled Offset/Rotate. For the second transmission profile the direction is zero. To understand how the direction imagine you have a stick that you place attached to start point of the transmission profile and parallel to the X axis, then you raise it the indicated angle. Thus to get the first transmission profile to go up, we need to point the stick the opposite direction of the X axis, hence 180 degrees.

8.6 A basin

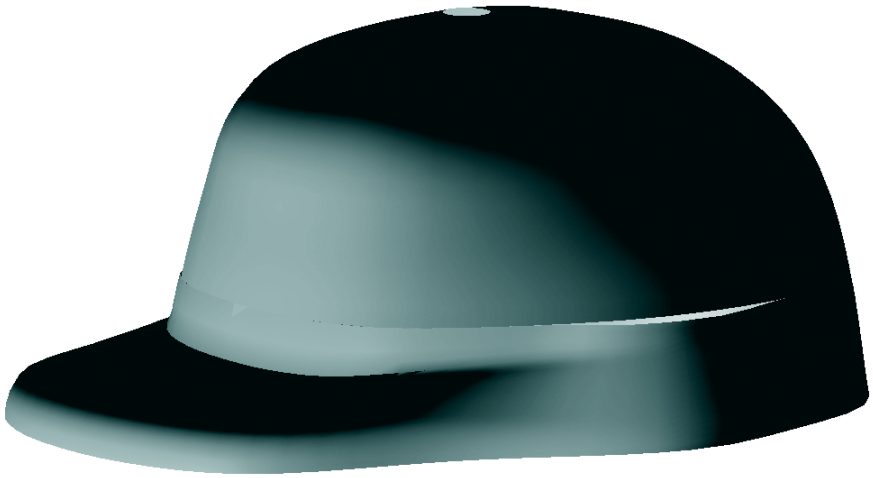


This drawing is a snake with 2 transmission profiles and 13 shift profiles. The transmission profiles are closed, i.e. their start and end points are connected. Therefore we need to tell which way to go around. The direction is controlled by coloring of the first and last entities. The last entity must have a lower priority color than the start entity. For example if the start entity is black (highest priority) the end entity can have any non-gray-scale color, e.g. blue. To get to desired result the second transmission profile must be 44.7 mm lower than the first. To obtain this we must offset the 2nd transmission profile -44.7 in Z. This is done on the 3rd page of the snake process (see ?? of the previous lesson). In this case we enter -44.7 in the Z field of the 2nd transmission profile, all the other fields are left empty.

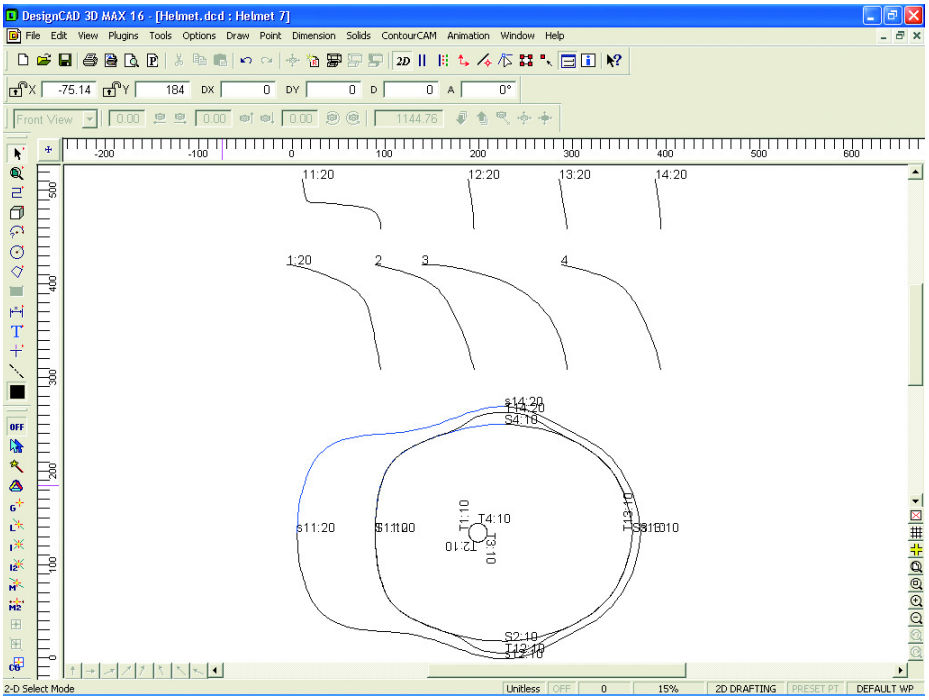

DesignCAD 3D MAX 16 - [A Basin 3D.dcd]
File Edit View Plugins Tools Options Draw Point Dimension Solids ContourCAM Animation Window Help
Perspective
A Basin 3D.dcd : front
A Basin 3D.dcd : top
A Basin 3D.dcd : side
A Basin 3D.dcd : main
2-D Select Mode
3D WIREFRAME
DEFAULT WP

The get a complete program you might want to do a pocket milling to clear the bottom. To make a pocket milling you will need to make a copy of the 2nd transmission profile to use as the border for the pocket milling.

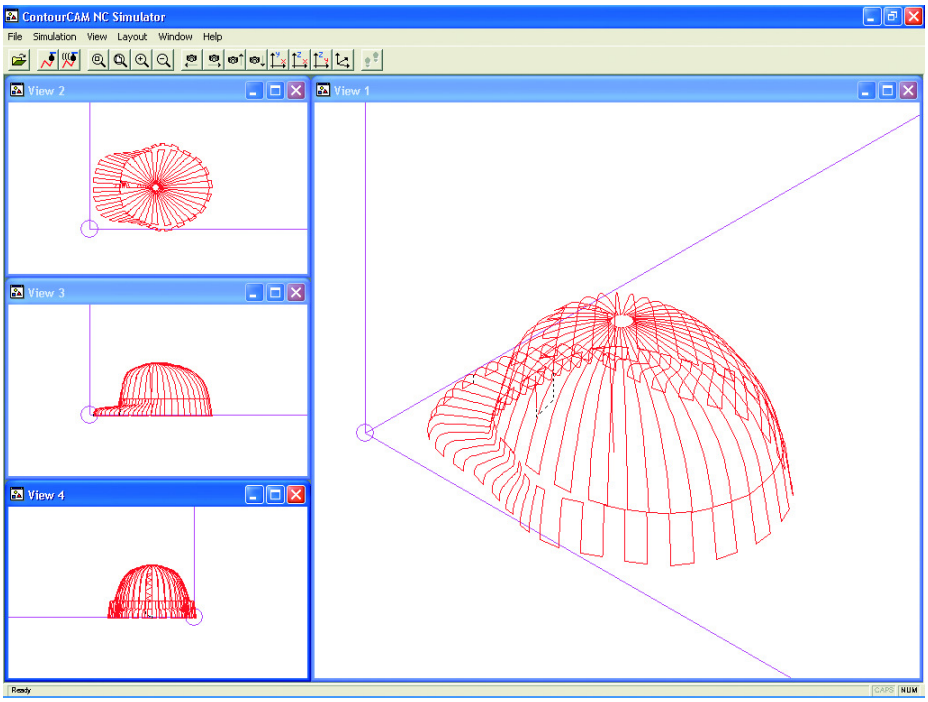
8.7 Helmet



In this lesson we will construct a helmet, as shown above. The program is made in DesignCAD 2D since all transmission and shift profiles are planar. To make the helmet we use two snake processes. The first forms the top to the ears and the other from the ears and down.

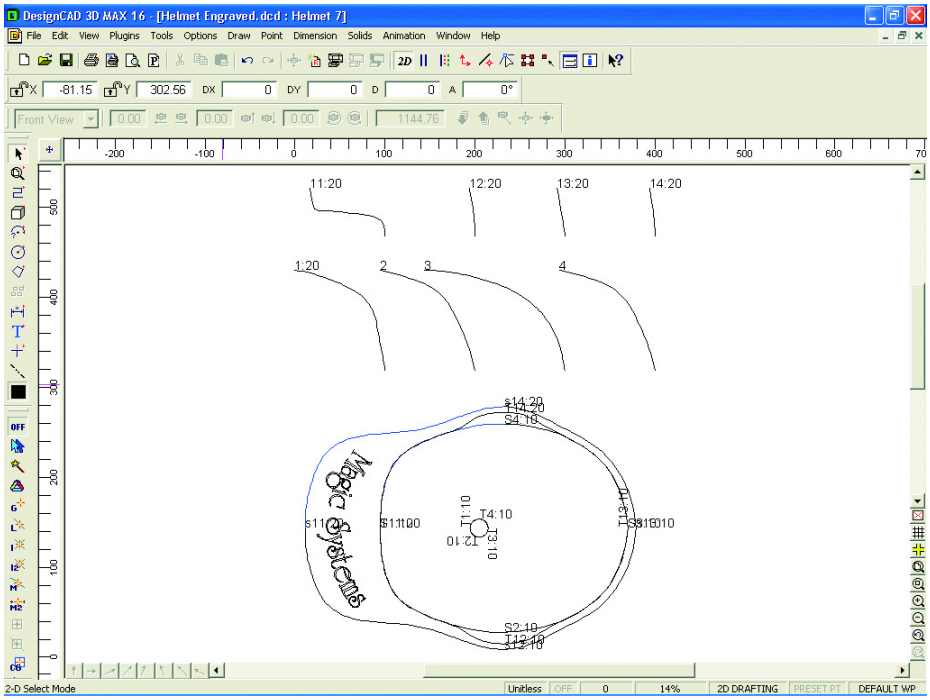


Profiles for the first snake process. The profiles for the first snake process are shown here. The easiest way to draw the profiles is by drawing curves and use the curve-to-line command to convert them to polylines. In the real life the profile might be digitized from a model. Draw all elements for the first snake process in layer 1. Remember that you will need to offset one of the transmission profiles to get it in the correct height. Observe that the 1st transmission profile is the same as the 2nd transmission profile for the 1st snake process except for some deviation by each ear. Make the drawing for the 2nd snake process in layer 2.

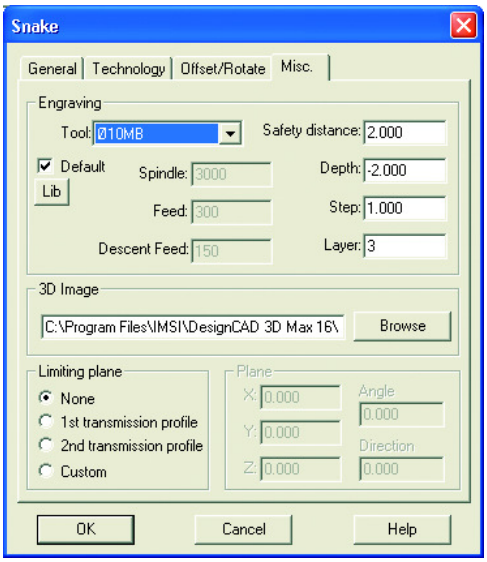

ContourCAM NC Simulator
File Simulation View Layout Window Help
View 2
View 3
View 4
View 1
Ready
CAPS NUM

8.8 Engraving in a Snake

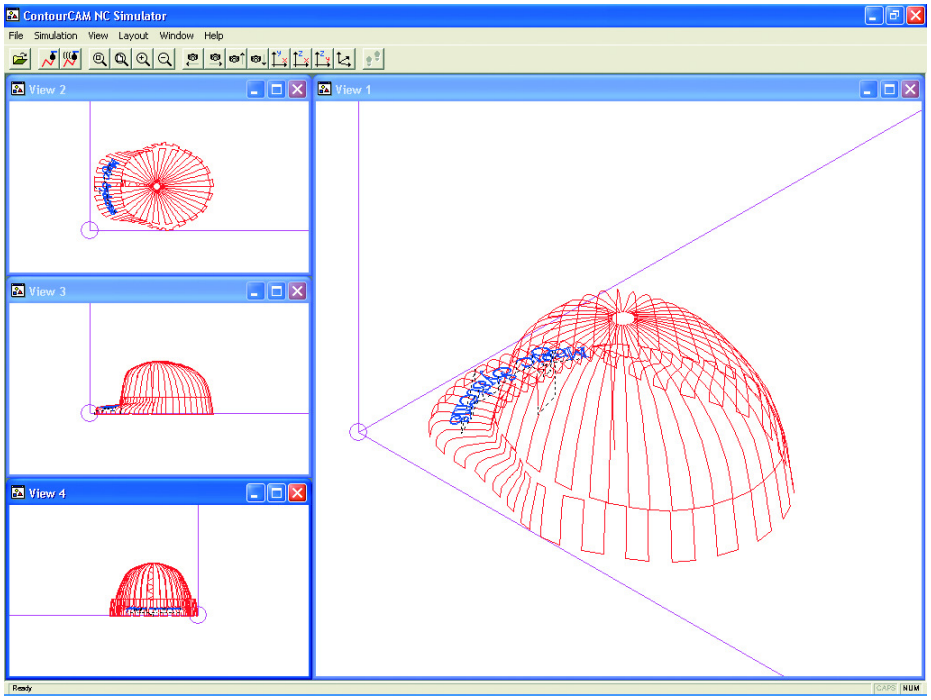
ContourCAM offers the possibility for engraving texts, logos etc. In surfaces defined by snakes. All text fonts can be engraved, but the text needs to be vector converted.



The text to engrave must be inside the area covered by the surface defined by the snake program. In this lesson the surface is inside a semi circle, thus the text must also be within this area. In might be advantageous to place the text etc. To be engraved in a separate layer, but it is not mandatory. If you to the elements to be engraved in the same layer as the snake, all that is not part of the snake definition will be engraved in the surface. In the snake process you should check the option engrave on the first page (General) to tell ContourCAM that you want to do an engraving.



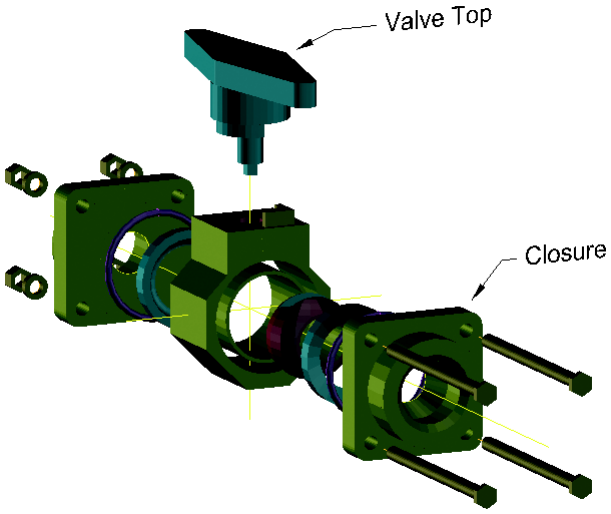
On the page labeled Engrave you specify the information needed to do the engraving, see ??. We enter 2 in Safety distance to allow the milling machine to descent with rapid traverse until 2 mm above the surface. In Depth we enter -3 to make the engraving 3 mm into the surface. The let the text etc. follow the surface we must divide all lines and arcs into small line segments. The field Step specifies how long these line segments should be.



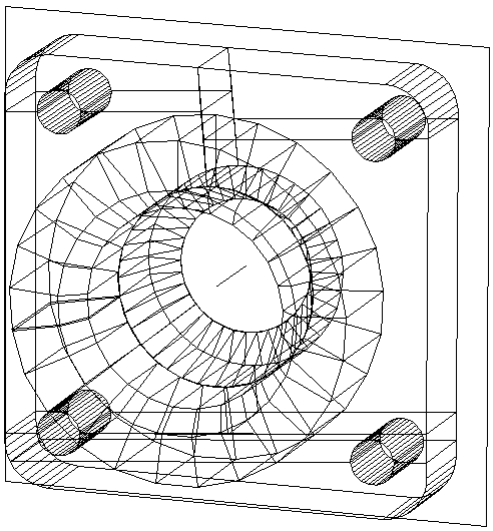
8.9 Carpet

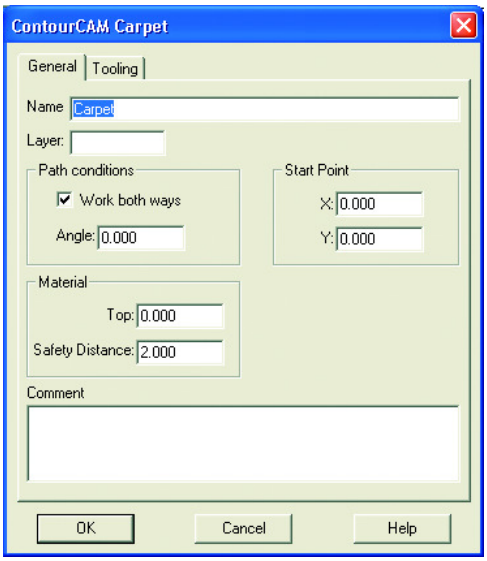
To show you how the carpet function works, we use one of the examples in the \Extended Samples, namely, some parts off the valve.

We select the closure and copy it to a new drawing.

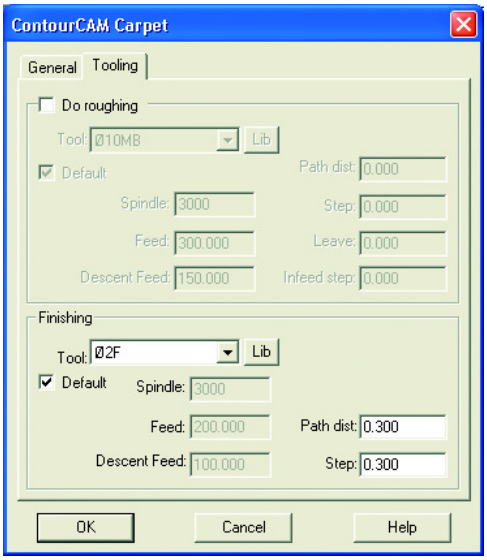


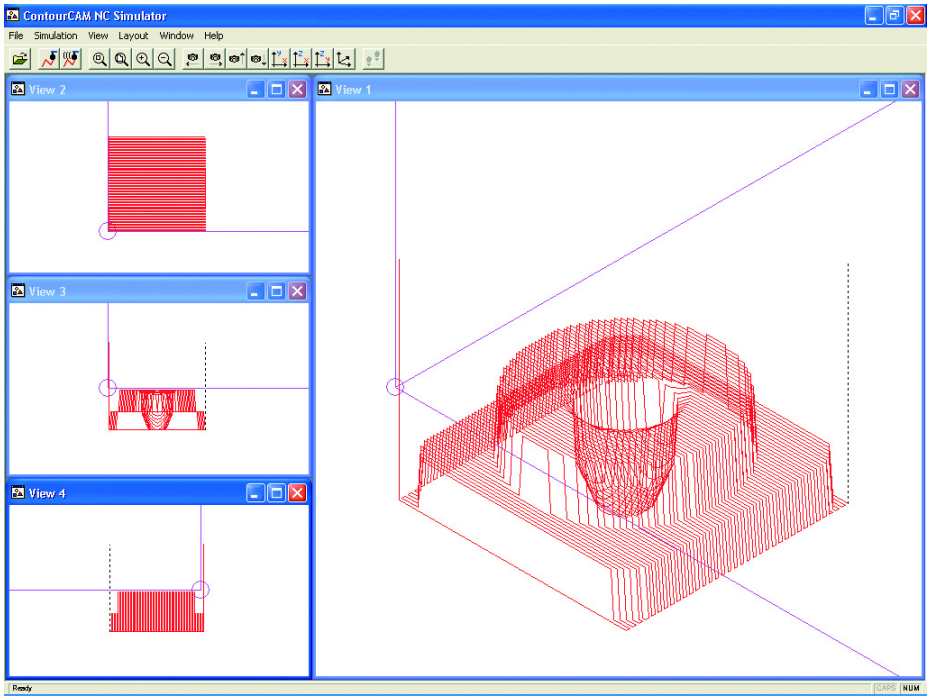
To be able to work with it in the correct plane, we need to rotate it and lay a plane underneath.





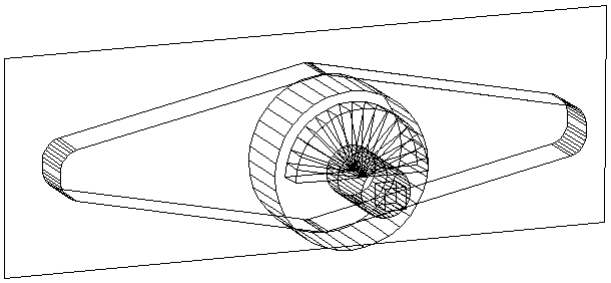
Then you select ContourCAM=> Milling=>Carpet and fill in the windows.

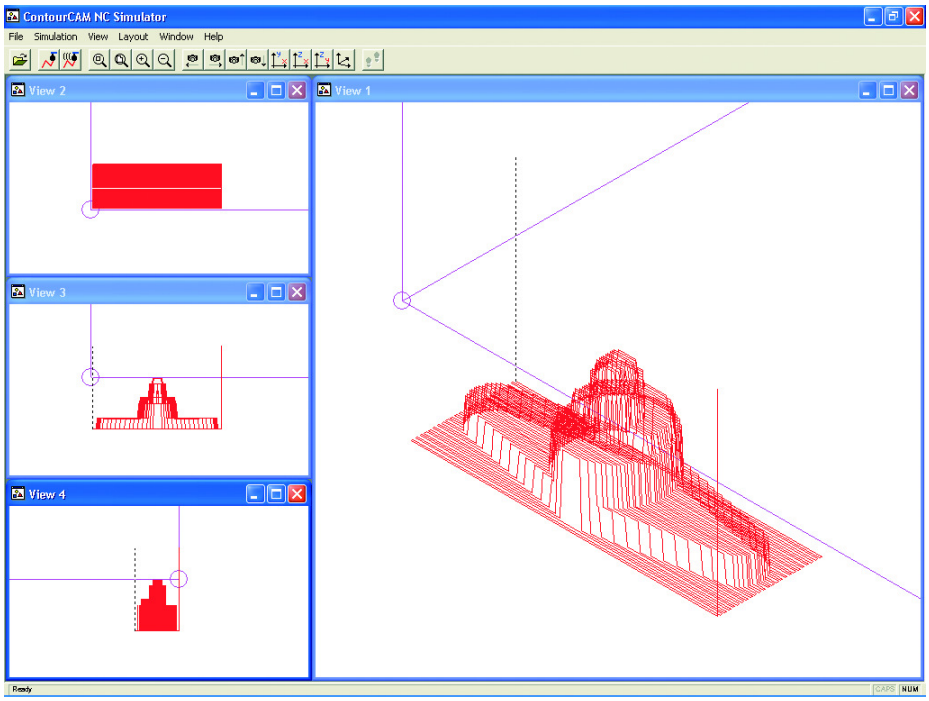




And the Carpet function does the rest.

So now we do the top of the valve the same way.




ContourCAM NC Simulator
File Simulation View Layout Window Help
View 2
View 3
View 4
View 1
Ready
CAPS NUM

Introduction

Getting Started

Introduction 1
System Requirements 1
Installing ContourCAM 2
Manually Installing ContourCAM 2
ContourCAM 2D and 3D 2
ContourCAM Setup 2
Sources of Information About ContourCAM 3
The User's Guide 3
The Online Help System 3
The Sample Drawings 3

Chapter 1 ContourCAM Configuration

1.1 General Settings 1-1
1.2 Default Values 1-2
1.3 Tool Library 1-2
1.3.1 Tools for milling machines 1-3
1.4 Machine Cycle Library 1-3
1.4.1 Editing machine cycles 1-4
1.5 Postprocessors 1-4
1.5.1 Postprocessor format 1-5
1.5.1.1 Format-tab 1-5
1.5.1.2 Coordinate-tab 1-6
1.5.1.3 Options-tab 1-7
1.5.1.4 Decimal-tab 1-8
1.5.2 Sequences in the postprocessor 1-9
1.5.2.1 Header-tab 1-10
1.5.2.2 Program-tab 1-10
1.5.2.3 Tool change 1-11
1.5.2.4 Misc.-tab 1-11
1.5.3 Settings 1-11
1.5.3.1 Files-tab 1-12
1.5.3.2 Serial-tab 1-12
1.5.3.3 Value-tab 1-14
1.5.4 Tool library 1-14
1.5.5 Machine cycles 1-15
1.6 Assign default postprocessor 1-16

Chapter 2 ContourCAM Editor

2.1 Editing files 2-1
2.1.1 Short-cuts while editing 2-2
2.1.2 Hot-keys in ContourCAM Editor 2-3
2.1.3 Include file 2-4

2.1.4 Include from serial 2-4
2.2 Search functions 2-4
2.3 CAM-functions 2-5
2.3.1 Choose postprocessor 2-5
2.3.2 Dimension offset 2-6
2.3.3 Remove Line Numbers 2-6
2.3.4 Add Line Numbers 2-6
2.3.5 Insert start-/stop sequences 2-6
2.3.6 Drilling on lathes 2-6
2.3.7 Mirror in X 2-7
2.3.7 Mirror in Y 2-7
2.3.9 Rotate 2-8
2.4 Calling external programs 2-8
2.4.1 Receie program 2-8
2.4.2 Send program 2-9
2.4.3 Send to file 2-9
2.4.4 Simulate NC program 2-9
2.5 Loop programming 2-9

Chapter 3 Serial Communication

3.1 Reception from a serial port 3-1
3.2 Transmission via a serial port 3-2
3.3 TRansmission to a file 3-2

Chapter 4 NC Simulator

Chapter 5 COntourCAM 2D

5.1 Editing commands 5-1
5.1.1 MabicTrim: Trimming more elements 5-2
5.1.2 Fillet-2: Draw an arc tangent to line/circle 5-2
5.1.3 FIllet to point: Draws an arc through a point tangent to a circle 5-3
5.1.4 Delete all but selected color 5-3
5.1.5 Delete but selected: Delete all except what is selected 5-4
5.2 Drawing commands 5-4
5.2.1 LinePDXY: Relative line 5-4
5.2.2 Box2: Draw squares 5-5
5.2.3 Spiral: DRaw a sprial 5-5
5.3 Nest commands 5-5
5.3.1 Nest-1: Nest one geometry to itself 5-5
5.3.2 Nest-2: Nest two geometries to each other 5-6
5.4 NC programming 5-6
5.4.1 Generating NC code from a drawning 5-7
5.4.1.1 Drawing format 5-7

5.4.1.2 Types of entities....5-7
5.4.1.3 Defing of toolpath....5-7
5.4.2 Working processes....5-8
5.4.2.1 Parameters....5-9
5.4.2.2 Generating of an NC program....5-9
5.5 Milling....5-9
5.5.1 Milling of a toolpath....5-10
5.5.2 Milling of contours....5-11
5.5.3 Pocket milling....5-12
5.5.4 Boring etc.....5-15
5.5.5 Engraving....5-17
5.6 Cutting....5-18
5.6.1 Cutting paths....5-18
5.6.2 Contour cutting....5-20
5.7 Turning....5-21
5.7.1 Contour turning....5-21

Chapter 6 ContourCAM 3D

6.1 Snake programming....6-2
6.1.1 Snake-1 programming from DesignCAD 2D....6-2
6.1.2 Basic snake programming....6-3
6.1.2.1 The shift profiles....6-3
6.1.2.2 The transmission profiles....6-4
6.1.3 The placements and limitations of the knots....6-5
6.1.4 Engraving a snake....6-6
6.1.5 Creating a snake process....6-6
6.2 Milling Carpet....6-11

Chapter 7 Tutorial ContourCAM 2D

7.1 Examples of drawing editing and nest commands....7-1
7.1.1 Fillet-2: Draw an arc between circle and line....7-1
7.1.2 Fillet-2 Draw an arc between two circles....7-2
7.1.3 Line PDXYDrawing line dialog....7-3
7.1.4 Delete all but specified color....7-4
7.1.5 MagicTrim: Possibilities, Example 1....7-4
7.1.6 MagicTrim: Possibilities, Example 2....7-5
7.1.7 Nest-1: Nest an item to itself....7-5
7.1.8 Nest-2: Nest two different items to each other....7-6
7.2 Defining tools and machine cycles....7-7
7.2.1 Definition of tools....7-7
7.2.2 Definition of drilling cycle....7-9
7.3 Examples for NC generation....7-10
7.3.1 Preparation of a drawing for NC generation....7-10
7.3.1.1 The construction phase....7-11
7.3.2 Generating the code....7-12
7.3.3 In ContourCAM's Editor....7-14

7.3.4 Cutting priorities generating NC-code 7-14
7.3.5 Pocket milling: Advanced 7-18
7.3.8 In the Editor of ContourCAM 7-20
7.3.9 Machine cycles..... 7-21

Chapter 8 Tutorial ContourCAM 3D

8.1 Your first snake 8-1
8.2 Using knots 8-4
8.3 ISO corner 8-6
8.4 Rotating Shift Profiles..... 8-8
8.5 Rotating the Transmission Profiles 8-9
8.6 A basin 8-10
8.7 Helmet 8-12
8.8 Engraving a Snake 8-15
8.9 Carpet 8-17